NASA Technical Memorandum 100790
ICOMP-88-1

Institute for Computational Mechanics in Propulsion (ICOMP)

Second Annual Report—1987
produced by the ICOMP Steering Committee

(NASA-TM-100790) INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP) Annual Report No. 2, 1987 (NASA) 49 p CSCL 12A

N88-19202

Unclas
G3/64 0130094

March 1988

Institute for Computational Mechanics in Propulsion (ICOMP)

Second Annual Report—1987

National Aeronautics and
Space Administration

CONTENTS

PRECEDING PAGE BLANK NOT FILMED

THE INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP)

SECOND ANNUAL REPORT

1987

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1987.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) is jointly operated at the NASA Lewis Research Center by Case Western Reserve University and NASA Lewis under a Space Act Agreement. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion. The organization and operation of ICOMP have been described in ICOMP Report No. 87-8 (NASA TM 100225), "The Institute for Computational Mechanics in Propulsion(ICOMP), First Annual Report," Nov., 1988, 14 pages.

The scope of the ICOMP research program is: to advance the understanding of aerospace propulsion physical phenomena; to improve computer simulation of aerospace propulsion components; and to focus interdisciplinary computational research efforts. The specific areas of interest in computational research include: fluid mechanics for internal flow; structural mechanics and dynamics; multivariable control theory and applications; and chemistry and material science.

The report summarizes the activities at ICOMP during 1987. It lists the visiting researchers, their affiliations and time of visit followed by reports of RESEARCH IN PROGRESS, REPORTS AND ABSTRACTS, and SEMINARS presented.

THE ICOMP STAFF OF VISITING RESEARCHERS

The composition of the ICOMP staff during 1987 is shown in figure 1. Forty-three researchers were in residence at Lewis for periods varying from a few days to a year. The number in residence on a weekly basis is shown in figure 2. This figure clearly shows the "high season" during the summer months when there was a peak of twenty-five in residence. Figure 3 is a photograph of the ICOMP Steering Committee and the visiting researchers, taken at a reception in June, 1987. Figure 4 lists the universities or other institutions represented and the number of people from each. The figure lists twenty-eight organizations. The next sections will describe the technical activities of the visiting researchers starting with reports of RESEARCH IN PROGRESS, followed by REPORTS AND ABSTRACTS, and finally, SEMINARS.

RESEARCH IN PROGRESS

FRED A. AKL, Ohio University

My research has been concerned with vibration analysis in parallel microcomputing architecture with the specific objective of establishing the computational characteristics associated with the mapping of the generalized eigensolution of linear elastic finite element models onto parallel binary tree computing architecture.

The method of recursive doubling has been applied to a set of N stiffness matrices of two-dimensional beam elements, where $N=2^n$ and n is positive integer. According to this method, if OP is an arbitrary associative operation that can be carried on the set of stiffness matrices [K], then serial implementation of the expression K_1 OP K_2 OP ... OP K_N requires N-1 steps while parallel implementation requires only $\log_2$ N parallel steps.

An occam parallel program has been successfully developed to run on a set of 2, 4, 8, 16, and 32 transputers mapped into a binary tree environment. A serial version of the same algorithm has also been implemented on the transputer development board to compare its performance with the parallel program. Preliminary results indicate that addition of stiffness matrices in a transputer network of binary trees can be efficient for the larger number of matrices. As expected, no significant improvement is observed for the case of 4 and 8 matrices.

This work will be continued to investigate the solution of the generalized eigen problem using a binary tree transputer architecture. The speed-up and efficiency of the binary tree architecture also will be studied.

S. M. ARNOLD, University of Akron

Structural alloys used in high temperature applications exhibit complex thermomechanical behavior that is inherently time-dependent and hereditary, in the sense that current behavior depends not only on current conditions but on thermomechanical history as well. Recent attention has been focused on metal-matrix composite materials for elevated temperature applications, e.g. aerospace, where they exhibit all the complexities of conventional alloys (creep, relaxation, recovery, rate sensitivity, etc.) and their strong anisotropy adds further complexities.

One approach in representing the deformation behavior of metal-matrix composite materials is to view the composite (fiber-matrix combination) as a material in its own right, with its own properties that are measurable through appropriate phenomenological tests. Following this continuum viewpoint, high temperature deformation behavior of metallic composites can be idealized as pseudohomogeneous continua with locally definable directional characteristics. Pseudohomogenization of textured materials and applicability of continuum mechanics depends relatively upon characteristic structural dimensions, severity of gradients and the size of the internal structure of the material. Examination reveals that the appropriate conditions are met in a sufficiently large class of anticipated applications of metallic composites to justify research into the formulation of continuum based theories.

In recent months the author's research has been in support of this continuum viewpoint, specifically with respect to an examination of the fabrication process for tubular composite test specimens and in the development of a nonisothermal transversely isotropic elasticity theory. The fabrication process is being addressed, because a prerequisite in adopting the continuum approach is that reasonable integrity and reproducibility of material properties can be maintained even under somewhat varying fabrication processes as may be required for manufacturing various component or specimen configurations.

A qualitative examination of the fabrication process for a tubular tungsten fiber reinforced Kanthal (W/Kanthal) composite test specimen has been completed. The analysis consisted of idealizing the tubular specimen as a system of three perfectly bonded concentric thick-walled cylinders. Each cylinder was assumed to behave linearly elastic throughout the entire process. Results of the study indicated the potential for cracking of the tungsten fibers under the current fabrication process and are in agreement with metallographic observations. Details of this study and possible guidelines for modification of the current fabrication process to reduce/avoid fiber cracking are presented in a technical report (number yet to be assigned).

Presently a nonisothermal transversely isotropic (a single family of fibers with a preferred direction) elasticity theory is being developed. This theory, in conjunction with the extension of the isothermal transversely isotropic viscoplastic theory put forth by Robinson, will provide a consistent thermomechanical deformation theory for metallic composites composed of a single family of fibers. The present elastic theory is consistent with Robinson's viscoplastic theory in that the directional dependence of the composite is incorporated into the potential function, albeit strain energy or complementary energy, from the onset. Details of this theory will be ready for release shortly. Also, it is anticipated that in the near future this theory will be coupled with an anisotropic creep damage model.

R. J. BODONYI, Ohio State University

Branching and merging flows are of interest in practical terms in internal flows, and they are also of interest more generally with regard to the understanding of fundamental fluid dynamics since they can contain some or all of the elements of separation, eddies, reattachment, three-dimensionality, trailing-edge and leading-edge properties. Basic to the understanding of these complicated features and their influence, however, is the resolution first of more central and model problems which clarify the nature of a few of the possible main features. With this goal in mind, a numerical study of the unsteady pulsating divided flow field produced when the flow in a two-dimensional channel is split into two equal parallel channels, separated downstream by an aligned splitter plate was initiated. Specifically, the two-dimensional incompressible Navier-Stokes equations, using a vorticity-stream function formulation, have been studied for the flow generated by an oscillatory flow upstream of the splitter plate. Since the fluid motion is forced by a purely harmonic motion upstream, it is assumed that the solution of the problem is also periodic, and so the solution can be decomposed in time using a Fourier series. This approach is thought to have a number of advantages, such as (1) an automatic "building in" of the expected periodicity of the solution, bypassing any transient solutions that may exist and (2) giving a better insight into the different components of the solution, in particular, the steady streaming component of the solution.

During this author's visit to the ICOMP Program, a numerical code for solving the appropriate system of partial differential equations has been developed. Initial testing of the code for several values of the two parameters appearing in the boundary-value problem, namely the Reynolds number and the Strouhal number, is underway. Preliminary results have been encouraging and a detailed study of the solution for a range of values of the Reynolds and Strouhal numbers is currently under investigation.

EDWARD A. BOGUCZ, Syracuse University

During my 8-week appointment at ICOMP, I made progress on two research projects. The first concerned unsteady stagnation-point heat transfer due to the motion of freestream vortices[1]. The effects of freestream vortices on the flow and heat transfer in a stagnation-point boundary layer were considered for the case of a circular cylinder exposed to an approaching pair of counter-rotating line vortices imbedded in an otherwise uniform crossflow. A numerical solution for the unsteady stagnation-point flow was formulated, and results were calculated for several vortex flow situations. Perturbations in stagnation-point heat transfer of ±40% were found to be induced by vortex pairs that have strength and separation distance suggested by recent measurements and current LeRC experiments. In addition, separated flow in the boundary layer at the stagnation point was found to be produced for several cases of practical interest. The results suggest that the effects of freestream vortices may be significant in situations of current engineering interest, such as the interaction of unsteady wakes with downstream turbomachinery blades.

The second concerned turbulent wake calculations[2]. Numerical solutions of the boundary-layer equations were obtained for the symmetric turbulent wake of a thin flat plate. In the near wake, a simple algebraic eddy viscosity model developed in a recent analytical study was used, and the computed results were found to support the analytical description of the near wake flow, which is based on an asymptotic analysis of the governing equations. Downstream of the near-wake region, accurate representations of experimental data were obtained using an algebraic model in which the eddy viscosity is constant across the wake, but linearly increasing with distance from the trailing edge. The computed results suggest further analytical work, which is currently underway.

NOTES

[1]Work pursued in collaboration with F. A. Lyman, who initiated a study of unsteady boundary-layer heat transfer due to the motion of freestream vortices during an appointment at ICOMP from September, 1985 to May, 1986.

[2]Work pursued in collaboration with J. D. A. Walker during his 4-week stay at ICOMP in June/July, 1987.

LOLA BOYCE, The University of Texas at San Antonio

Probabilistic methods are particularly useful in the design and analysis of critical systems and components that operate in severe and uncertain environments. These methods have recently found application in space propulsion systems to improve the reliability of engine components. In order to address the reliability of such components, material strength degradation as well as loading and structural analysis must be addressed in a probabilistic manner.

Material strength degradation is quantified by damage indicators such as crack length, fatigue strength or stiffness degradation, that change over time. Usually experimental data that relate damage and time are represented as deterministic mathematical models using methods such as fracture mechanics, micromechanics constitutive theory or simply a power law representation of the fatigue S-N diagram. Models may include parameter effects such as temperature, casting porosity, mean stress, residual stresses, and so on. They can be used to predict the "time to reach a critical material damage level." Thus, aerospace propulsion system component lifetimes may be estimated.

Experience has shown that under fatigue loading, such estimates only roughly correlate with widely scattered observed lifetimes, since fatigue is essentially a random phenomenon. The probabilistic analysis of two selected material strength degradation models, a strength reduction model and a fatigue crack growth model, are carried out using simulation. The strength reduction model is subjected to two methods of probabilistic density function generation, namely, maximum penalized likelihood and maximum entropy. These probabilistic models predict the "random time to reach a critical material damage level." Thus, the information available upon which to estimate the lifetime of an aerospace propulsion system component is increased.

These probabilistic methodologies are embodied in three algorithms and coded in FORTRAN. Example problems that consider data approximately typical of cast nickel base super alloys are presented. Results are presented in the form of probability density functions and cumulative distribution functions of the random time (in log of cycles) to reach a critical crack size or a critical alternating stress level, the fatigue strength.

FRED SHIH-HUNG CHANG, Cleveland State University

The research on developing uniform order essentially non-oscillatory (ENO) schemes was continued. Algorithms have been developed, coded and tested on model problems. First, the numerical experiment was done on single conservation laws, with both convex and non-convex cases, and very good results were obtained. Then a uniform second-order ENO algorithm was developed for the 1D Euler equations. In this algorithm, an approximate Riemann solver of Roe's type was used. The testing results on Sod's and Lax's problems are very good as expected. An exact Riemann solver was then developed for the 1D Euler equations. Very encouraging results were obtained on a very strong shock problem. An extensive study on the comparison with TVD schemes is being made in cooperation with Dr. M. S. Liou of the CFD Branch at NASA Lewis.

A two-dimensional version of the ENO scheme based on the two-dimensional Euler equations was then developed. The numerical computation was performed on the problem involving the reflection of a shock from a plane wall. The upstream Mach number was 2.9 and shock angle 29 degrees. The two-dimensional ENO results are very good and prove that this approach to shock capturing is indeed very promising.

TIAO J. CHANG, Ohio University

A seminar entitled, "Stochastic Approach for Studying Occurrences of Rare Events," was presented. The developed nonhomogeneous Poisson process for occurrences of rare events can be used to investigate the behaviors of airplane accidents due to failures of certain type of engine or of a particular design. Alternatively, a deterministic approach has been used for the study of air pressure-surge phenomena in a supersonic inlet.

The optimum design of a supersonic inlet and its associated auxiliary air systems must supply a continuous airflow required by the engine at certain pressure level. Due to atmospheric turbulence or acoustical resonance, large inlet flow fluctuations may occur and result in undesirable pressure surges that damage inlet structures and their auxiliary systems. Therefore, it is desired to precisely understand the air pressure-surge phenomena for designing aircraft inlet systems. Prior difficulties in studying the air pressure-surge phenomena have centered around the discontinuous problems of numerical solutions to the governing equations. Methods using the lumped volume scheme approximate transient flows by transfer functions to simulate pressure signal transit times. These methods may be sufficiently accurate for a short air induction duct, but their accuracies deteriorate for a long inlet system. The linearized small perturbation method, assuming that the dependent variables deviate only slightly from their steady-state values, combines a set of linearized equations with an exact solution of linearized wave equations to solve the problem. Since the method is based on a linearized treatment of equations, it is strictly applicable only for an inlet with small flow perturbations. The method of characteristics arranges the governing equations into the characteristic forms and solves the problems iteratively along the characteristic lines. However, the discontinuities due to shock waves will appreciably complicate the programming requirements of the scheme. The objective of this proposed study is to develop the governing equations for the air pressure-surge phenomena and to solve these equations by the total variation diminishing scheme to obtain a complete understanding of pressure-surge effects on inlet system designs. The total variation diminishing scheme has been demonstrated successfully in solving the discontinuity problems of numerical solutions by limiting antidiffusive flux differences with some nonlinear functions.

ABHISAK CHULYA, Cleveland State University

My research involves the development of fast numerical integration techniques for the stiff differential equations that govern the constitutive behavior of materials operating at high temperatures. These equations involve the coupled plasticity/creep response of materials to thermomechanical loads. They constitute the major difficulty in the economic computer simulation of the structural response of aerospace propulsion machinery operating at high temperatures.

STEPHEN COWLEY, Imperial College of Science and Technology

Work by Professor F. T. Smith and others has shown that, in the limit of high Reynolds numbers, Tollmien-Schlichting (T-S) waves can be described by the "triple-deck" boundary-layer equations. To be more precise, T-S waves can be modelled by these equations in the region of parameter space near the lower branch of the linear neutral curve. However, T-S waves are not the only instabilities that these equations describe; Tutty and Cowley (1986) and Smith and Bodonyi (1985) have shown that Rayleigh waves are another possibility. Tutty and Cowley (1986) showed that the Rayleigh instability leaves the triple-deck problem ill-posed, with the rate-of-growth of the instability increasing monotonically with wavenumber (cf. Kelvin-Helmholtz instability in inviscid flow). In order to obtain accurate numerical solutions it is therefore necessary to control rounding error. The governing equations were solved by a special Fourier method incorporating the filtering technique proposed by Krasny (1986) for the Kelvin-Helmholtz problem. This "cut-off" filter consists of setting all Fourier modes with magnitude less than a rounding constant to zero. As a less crude filter, an attempt was also made to model the expected exponential tail of the Fourier series which are used to describe the physical quantities of interest. It was shown by an analysis of this exponential decay that a singularity could develop after a finite time; this was indicated by the Fourier modes decaying algebraically, rather than exponentially, with wavenumber at this time. The singularity indicates that the model of T-S waves using the triple-deck equations is only valid for a finite time. Although the calculations were done for two-dimensional flows, similar instabilities also occur in three-dimensions, as is more relevant for "transition to turbulence."

REFERENCES

Krasny, R. (1986) J. Fluid Mech. 167, pp 65-93

Smith, F. T. and Bodonyi, R. J. (1985) Aero. J. 89, pp 205-212

Tutty, O. R. and Cowley, S. J. (1986) J. Fluid Mech. 168, pp 413-456

GEORGE S. DULIKRAVICH, Pennsylvania State University

I performed research on three different topics. The first topic involved development of a computer code for the analysis of given cascade shapes using Stream-Function-Coordinate (SFC) formulation for inviscid steady compressible flows. In addition, the same code is capable of iteratively determining shapes of the airfoils that correspond to the prescribed surface Mach number distribution.

The second project involved derivation of canonical forms of the SFC equations that are suitable for the aerodynamic analysis and inverse design of three-dimensional configurations.

The third project dealt with the numerical integration of one dimensional forms of the full potential transonic equation with different forms of numerical dissipation. A new form of the physically based numerical dissipation was tested and shown to be able to produce non-negative entropy jumps across shock waves. Moreover, the new formulation is entirely analytic. It is capable of producing shock waves of different strengths in a fully controlled fashion since the input parameters are physical coefficients of viscosity.

GRAHAM K. ELLIS, Virginia Polytechnic Institute and State University

The main goal of the research effort this past year has been the development of graphics and animation routines on the transputer based parallel processing system. Due to the immature software for this new system, considerable effort has gone into developing useful routines to supplement the development tools provided with the system.

Initial efforts concerned the development of a PC based graphics emulator so transputer graphics code could be developed without a transputer graphics board. A single transputer on a PC plug-in card was used for this research.

The Inmos Transputer Evaluation Module (ITEM) was received 6/87 and was installed and tested. Vendor-provided checkout software was used to perform this task.

The ITEM consists of 40 transputers with 256 KBytes of memory per processor. Additional hardware received includes a transputer based graphics board and 27 special integer transputers.

Due to the unusual software development environment used on the transputer, special disk input/output (I/O) routines were developed to provide access to stored files in a standard fashion. These routines provide a file interface similar to those in programming languages like C or FORTRAN.

One of the goals was to be able to plot (and animate) the output from a NASTRAN finite element analysis. A general purpose package consisting of two programs was written to perform data reduction and sorting on a NASTRAN data deck echo. Both programs were written in C because of its excellent string and test manipulation abilities as well as it's inherent portability from one computer system to another. All of the functions in the code are designed to be small, modular, and portable so users can quickly adapt the code to their particular needs.

A number of general two-dimensional graphics routines have been written to allow a user to generate graphics output from data provided to the transputer system. Standard windowing, clipping, line, polygon, circle and arc routines are provided to allow a user to work in a world coordinate system without concern for the graphics device being used for output.

Because the standard graphics board used on the transputer system is too slow for realistic animation, a revised version of the software drivers are being worked on to accelerate plotting speeds. The current research used 18 of the special integer transputers for the display processing rather than the standard single transputer. This work is currently in progress.

A side benefit of the multiple transputer display processor work has been the development of a method to simulate global memory on a transputer network with only local memory.

Other activities performed this year include serving as a technical advisor to summer faculty involved in transputer based research activities.

M. R. FOSTER, Ohio State University

I had in mind for my four-week stay at ICOMP that I would finish up some numerical work already begun using the CRAY to do some heavy computations. The research has to do with the stability of Long's vortex, a particular limit solution of the Navier-Stokes equations. The vortices are highly unstable to small, helical disturbances. Professor F. T. Smith of University College, London and I have completed an asymptotic analysis for both modes of the vortex; the sinuous and varicose modes. Numerical computation of these modes is accomplished by a Newton iteration procedure on vortex profiles obtained by a shooting technique. The results of the computations show good agreement with the asymptotic results for the sinuous mode, but the varicose mode numerics still strongly disagree. Since I have been here, the problem is a "mode-jumping" phenomenon, which has proved difficult to counter.

Several persons in ICOMP and also some permanent Lewis employees have offered helpful suggestions. Dr. Bodonyi, in particular, made a suggestion of an alteration of the scheme, which was easy to implement. Unfortunately, that did not solve the problem. Dr. Jacqmin made two observations about alternatives to the present procedure, one involving a variant numerical formulation, which I have not yet had the opportunity to implement. Dr. Goldstein and members of his group, in particular Drs. Lieb and Ashpis have offered some helpful comments on the course of the analysis.

In summary, in terms of helpful suggestions on the future course of my research from persons working at Lewis or at ICOMP and general interaction with fellow professionals, I have found this time to have been a most stimulating four weeks.

MURLI M. GUPTA, George Washington University

I worked on iterative methods for the solution of linear algebraic systems. If convection-diffusion equations are discretized using central differences and the resulting systems of algebraic equations are solved by point Jacobi method, the iterations diverge whenever the convection terms are large. The eigenvalue spectrum of the Jacobi iteration matrix has the property that the real part of all eigenvalues lie in (-1,1), though the imaginary parts can be very large. In such a case, a spectrum enveloping technique can be applied to obtain convergent iterations [1]. When Gauss-Seidel or successive over-relaxation is used to solve the linear system, the spectrum enveloping is not applicable as the real parts of many of the eigenvalues of the iteration matrix are smaller than -1. A scaling parameter can be defined in this case to transform the iteration matrix such that the real parts of all eigenvalues of the transformed matrix lie in (-1,1) and results of [1] are applicable.

I modified the spectrum enveloping algorithm of [1] to incorporate this spectrum transformation. The resulting code automatically locates the eigenvalue spectrum, estimates the parameters and decides whether spectrum transformation, spectrum enveloping or a combination are to be used. This code works well on convection-diffusion and biharmonic equations and is being tested for robustness and versatility on Amdahl 5860 and Cray X-MP24. Applications to Navier-Stokes equations are contemplated.

After attending one of the ICOMP seminars, I decided to take a look at the biharmonic equation. It is well known that the 13-point approximation leads to an ill-conditioned system; one encounters divergence when the Jacobi method is used to solve this system. I discovered that the real part of several of the eigenvalues of the Jacobi iteration matrix has a value close to -2. The matrix transformation technique described earlier is applicable and works very well. A surprising discovery was that the Gauss-Seidel iteration for the discrete biharmonic equation is convergent and the spectrum enveloping technique [1] is applicable for accelerating the convergence in this case. The code described earlier works very well for these problems. Applications to Stokes flow problems are contemplated.

REFERENCE

[1] Murli M. Gupta, SIAM J. Algebraic and Discrete Methods, Vol. 7 (1986), pp. 513-526.

THOMAS HAGSTROM, State University of New York at Stony Brook

The theme of my research at ICOMP was the development and testing of accurate boundary conditions to be imposed at artificial boundaries for the numerical solution of time dependent problems in fluid mechanics. Although the approach I used is a general one, my efforts were concentrated on two particular situations: the outflow boundary for a viscous, parallel shear flow and "radiation" conditions for exterior flows in isentropic gas dynamics.

The boundary conditions I propose for shear flows are based on the numerical solution of the spatial Orr-Sommerfield equation. Both the group velocity and the spatial exponential decay rate are computed. The latter, of course, is present due to the viscosity. To compute asymptotic expansions far downstream, I simply locate (numerically) the wave packets with the minimal spatial decay rates. Given these, accurate boundary conditions can be constructed (at least in the linear regime). For channel flows at moderate Reynolds numbers (say 100 to 4000), the dominant wave packets have temporal frequency zero - i.e. slowly varying solutions. For higher Reynolds number, a packet at a higher frequency comes into play. This is the wave which will eventually become unstable. Successful numerical tests of these conditions have been carried out at moderate Reynolds numbers for channel flows with imposed perturbations at inflow and are under way for flow over a backward facing step.

A future research project which will make use of these results is the study of the spatial development of subcritical and supercritical transitional flows. In the first place, the dispersion relations I have computed are useful in the development of analytical theories. Secondly, one expects the accurate treatment of the outflow boundary condition to be necessary for reliable long time computations, especially for an unstable flow.

The solution of the gas dynamics problem, done in conjunction with Professor S. I. Hariharan of the University of Akron, makes use of entirely different techniques. We assume the Euler equations govern the flow and that the far field solution is spherically symmetric. Many authors construct conditions based on a principle of no reflection - i.e. no incoming waves at the artificial boundary. It has been documented, however, that these yield disappointing results for problems with spherical symmetry. This is due to the fact that the incoming and outgoing waves are coupled - causing "natural" reflections. Our approach is to use the method of characteristics to compute an asymptotic solution of an appropriate initial boundary value problem in the region exterior to the computational domain. We find an asymptotic "reflection" operator and, hence, a reflecting boundary condition. We tested the condition for a bursting balloon problem and found that the results compared quite favorably to those produced with the non-reflecting condition. Even on small domains, the correct steady state was reached.

Among the directions for future work are the formal inclusion of deviations from spherical symmetry in the asymptotic analysis, testing of the conditions in a nonsymmetric case and applications to problems of physical interest.

S. I. HARAHARAN, University of Akron

One research problem of interest is the eddy-current situation that arises when an electromagnetic field is incident on a metallic region. Calculation of fields around and inside the metallic region is the goal of the research. For time harmonic fields with single angular frequency, substantial progress has been made in the past using the boundary integral equation (bie) method. We are investigating transient incident fields where the problem cannot be solved by bie. The work is done jointly with Dr. N. Ida (Electrical Engineering Department/University of Akron) and a graduate student, M. E. Lee.

This second research problem concentrates on two aspects of the work carried out recently by Dr. H. Atassi (Notre Dame) and J. Scott (NASA LeRC). First, the approach here will be to consider the problem in the time domain rather than in the frequency domain. By the principle of "limiting amplitude" in the presence of a periodic gust the solution will become time periodic and the harmonic steady state will be reached. Such a treatment allows calculations of the solution with an explicit finite difference scheme. Usually they are inexpensive over the implicit methods. The second aspect is that once the explicit method is verified, the procedure is readily available for a transient gust. This aspect of the research is still an open area. The work is being done in conjunction with J. Scott and Dr. H. Atassi.

BO-NAN JIANG, University of Texas

Finite element methods are flexible for handling complicated flow domains by using unstructured grids. This means that adaptive mesh refinement procedures can be conveniently incorporated. Application of these methods to solving various problems of fluid dynamics has been proven successful. Recently, much attention is being drawn to the development of finite element algorithms for the analysis of high speed compressible flows especially for treating discontinuities. Although significant progress has been achieved by using methods such as the Taylor-Galerkin, Petrov-Galerkin and least squares methods, much more development is needed to make these methods efficient and accurate for solving engineering problems.

At this moment, the Taylor-Galerkin method is the most popular because of its generality. However, the Taylor-Galerkin method has some weakness: (1) the explicit Taylor-Galerkin schemes suffer from severe stability limitations (Courant number $\pm 1/\sqrt{3}$,); (2) an implicit Tayler-Galerkin scheme can be constructed, but leads to inversion of a large nonsymmetric matrix;

(3) the Taylor-Galerkin method is essentially a high-order scheme, which produces oscillations. So far, the Taylor-Galerkin method must rely on the use of artificial diffusion to remove the oscillations. However, the form of this added term is not unique and the associated parameters must be controlled. The Petrov-Galerkin method seems more promising, but it may have the same problems [(1) and (2)] as the Taylor-Galerkin method.

For the numerical solution of two-dimensional compressible Euler equations, we proposed a class of least-squares methods. We begin by considering the first-order implicit time-differenced non-conservative formulation. The least-squares method is employed to minimize the residual in the L^2 norm. It was demonstrated that for first-order hyperbolic equations this approach yields a weak statement similar to streamline upwinding, but the matrix is symmetric. The associated artificial viscosity appears naturally and is dependent on time step Δt, but with no free parameters. For a linear problem, the implicit scheme is unconditionally stable for all Courant numbers.

The application of this L^2 method with linear elements to the compressible Euler equation produces non-oscillatory shock profiles. While almost all workers in finite element area use only linear elements, we also investigated the possibility of utilizing another great inherent strength of finite elements, i.e. the systematic use of high-order approximation. For high-order elements, in some cases the L^2 method produces non-physical oscillations. In order to overcome this difficulty, we proposed a new least-squares variational method based on minimizing the residual in an approximate H^1 norm. This strategy can be interpreted as a multi-objective programming technique which minimizes the residuals as well as the derivatives of the residual.

The distinguishing features of our method are: (a) unconditional stability; (2) symmetric matrix; (3) fast convergence to the steady state; (4) non-oscillatory shock profiles without any added dissipation for linear elements.

A study on converting the original nonconservative formulation into the conservative form is being made in cooperation with Dr. M.-S. Liou of the NASA Lewis Computational Fluid Dynamics Branch. The preliminary results on the shock reflection problem show that there is no significant difference between the conservative and the nonconservative results for steady state solutions. Further comparisons are being made.

KAROL Z. KORCZAK, Case Western Reserve University

The isoparametric spectral element method [1] is a blend of finite element philosophy and spectral methodology. The idea of combining the geometrical flexibility of finite elements and the high accuracy with negligible diffusion and dispersion errors of spectral techniques has been in the forefront of research that was sprung by the advances in computer technology, particularly vector architecture and parallel processing.

In the traditional finite element approach [2] the geometry is divided into large number of small subdomains (elements) and all functions are represented by low order (typically linear) polynomial expansions. An increase in accuracy is obtained by increasing the number of elements. This methodology results in algebraic convergence and substantial errors of diffusive and dispersive character. These problems have been recognized and gave rise to the p-type finite element methods [3] in which higher order expansions are used.

On the other hand, the global (spectral) methods [4] treat the domain as a whole incorporating high order expansions. An increase in accuracy is obtained by increasing the order of expansion. With a proper choice of functions, a rapid (exponential) convergence is achieved with negligible diffusion and dispersion errors. These properties are extremely important, particularly in unsteady complex flows. The main drawback associated with these techniques is the close dependence of the convergence rate on the smoothness of all functions involved. In case of complex geometries, geometrical singularities, discontinuous boundary conditions and some other situations, a global expansion is inappropriate and leads to unacceptable results.

A possible remedy in the above cases could be a spectral multidomain (subdomain) approach in which the domain is divided into relatively few subdomains treated as separate spectral domains with appropriate matching or patching along common boundaries. One of those, the global element method [5], allowed different order of expansions in elements, however, it resulted in very complex mathematical description. Among all those methods up to date, the most successful approach is the isoparametric spectral element approach [1] that proved to perform well in solutions to complex problems in fluid mechanics, heat transfer and other similar applications.

The isoparametric spectral element method uses finite element methodology on the global level (interactions between elements) and a spectral approach on the local level (interactions within each element). This arrangement allows efficient use of benefits associated with spectral methods by choosing the elements such that all possible discontinuities are located on elements' boundaries. For practical purposes, Lagrangian interpolants are incorporated permitting straightforward treatment of various boundary conditions and allowing all functions to be in real space. The functions expansions are, according to the isoparametric recipe, all of the same order and are based on Chebyshev or Legendre polynomials through Gauss-Lobatto Chebyshev or Legendre collocation points. The choice of expansions depends on a particular application with a slight preference for Legendre polynomials.

The concept of implementing the spectral element formulation to solvers for the compressible general Navier-Stokes equations was developed many months ago. The programming and testing have started during tenure at ICOMP and will continue with numerous tests, modifications and expansions in the years to come.

The developed algorithm treats the general, compressible, time-dependent Navier-Stokes Equations [6] using the spectral element formulation in space and high-order explicit schemes in time. Several algorithms for time integration [7] have been implemented and initial tests have been performed.

A significant part of the research effort has been devoted to developments of algorithms for mesh generation, formation of supplementary and control arrays and other essential details for proper incorporation of all components of the program. Several routines have been directly transplanted from the existing incompressible version of the program with appropriate modifications. The main art of the new program consists of two solvers: one based on a predictor-corrector scheme and the other on a combination of Runge-Kutta (fourth order) and predictor-corrector (Hamming) schemes.

The new algorithm has been applied to flows in simple geometries and tested extensively. The tests indicate a significant importance of the procedure in imposing boundary conditions. In addition, the time integration requires a scheme with correction (iterative correl.) of the function values at every step. Various candidates are currently tested and will be implemented in the future.

The research conducted at ICOMP laid foundations for the new method. In the months to come the final algorithm will be assembled and tested on flow in a real device with experimental data available to compare the accuracy and flexibility. The results up to date indicate that the formulation has a potential to become as powerful in compressible flows as its predecessor in incompressible flows, the isoparametric spectral element method.

REFERENCES

1. Korczak K.Z., Patera A.T., "An Isoparametric Spectral Element Method for Solution of the Navier-Stokes Equations in Complex Geometry," JFM, Vol. 62, 1986, pp. 361-382.

2. Strang, G., Fix G.J., "An Analysis of the Finite Element Method," Prentice-Hall, Englewood Cliffs, N.J., 1973.

3. Babuska I., Dorr M.R., "Error Estimates for the Combined h and p Versions of the Finite Element Method," Numer. Math., Vol. 37, 1981.

4. Gottlieb D.O., Orszag S.A., "Numerical Analysis of Spectral Methods: Theory and Applications," SIAM, Philadelphia, 1977.

5. Delves L.M., Phillips C., "A Fast Implementation of the Global Element Method," J. Inst. Math. Applic., Vol. 25, 1980.

6. Anderson A.A., Tannehill J.C., Pletcher R.H., "Computational Fluid Mechanics and Heat Transfer," McGraw-Hill Book Co., 1984.

7. Carnahan B., Luther H.A., Wilkes J.O., "Applied Numerical Methods," John Wiley & Sons, 1969.

SHIH-TUEN LEE, National Taiwan University

A model for a turbulent diffusion flame adjacent to a solid fuel has been formulated. This model includes: (1) a low Reynold's number k-ε model, (2) an eddy-dissipation combustion model, (3) a soot formation and oxidation model and (4) a two flux radiation heat transfer model. Fast chemistry is assumed in the combustion model and in the soot oxidation model, but the soot formation model is based on finite-rate chemistry.

The resulting boundary layer equations have been solved numerically by a non-iterative marching scheme proposed by Patankar and Spalding. The results from the limited cases studied so far show that the radiation heat flux is quite important in a sooting flame.

In the future, the model will be improved by incorporating a finite-rate combustion model in order that the model can predict flame extinction properly.

B. P. LEONARD, University of Akron

The initial development stage of SHARP (Simple High Accuracy Resolution Program) was completed. This is an important enhancement of the widely used QUICK scheme, totally eliminating the unphysical overshoots to which QUICK is susceptible in modelling near-discontinuities, but without corrupting the global third-order accuracy. Results of this work were reported in the paper "Locally Modified QUICK Scheme for Highly Convective two- and three-dimensional Flows," presented at the Fifth International Conference on Numerical Methods in Laminar and Turbulent Flow, held at Montreal, Canada, July 6-10, 1987. SHARP uses the standard third-order accurate QUICK scheme in smooth regions, automatically switching to a more sophisticated form of flux modelling based on an Exponential Upwinding or Linear Extrapolation Refinement (EULER) in regions of rapid gradient variation (high local curvature). In principle, this is a cost-effective strategy; however, the current code used an explicit time-marching procedure to reach the steady-state solution, and the required small time step makes this an inefficient method for practical solutions. It is a straightforward matter to develop an implicit (ADI) solver based on the highly efficient pentadiagonal matrix algorithm (PDMA) thus producing, for the first time, an inherently multidimensional and non-oscillatory convection scheme for practical simulation of steady high-Reynolds-number flows containing near-discontinuities, such as thin boundary-layers, free shear-layers, shock waves, and various interactions, without the need for costly adaptive grid refinement. This forms the next logical step in the SHARP research project.

UTOPIA is a Uniformly Third Order Polynomial Interpolation Algorithm representing a two-dimensional (and three dimensional) extension of QUICKEST. Under pure constant- velocity convection conditions, a convected variable at time-level n+1 is equal to the value of that variable at time-level n at a distance $|v|\Delta t$ upstream. QUICKEST was developed from this idea in one dimension. UTOPIA is the two-dimensional or 3-D vector version. The resulting algorithm can always be written in a strictly conservative control-volume (CV) form; and it is a relatively straight-forward procedure to include physical diffusion terms in a consistent manner. Once the CV form has been developed, it is assumed that this is valid in the case of variable convecting velocities and diffusion coefficients. The resulting formulas for CV face convective fluxes involve the one-dimensional QUICKEST terms (including the important stabilizing normal gradient and normal curvature terms), a transverse gradient, a transverse curvature, and a twist term. The explicit update scheme is stable for Courant numbers of order unity, which is also appropriate for good time-resolution. A Fourier von Neumann analysis shows that the complex amplitude ratio matches that of the exact (convection-diffusion) equation through third-order terms in the dimensionless wave-number. In particular, leading order phase error is a very small fifth-order term; numerical dissipation is a small spatial fourth derivative; there is absolutely no "artificial numerical viscosity" (spatial second derivative). UTOPIA will thus provide a highly accurate and phase-stable unsteady code for multidimensional computation of very-high-Reynolds-number flows, such as large-eddy or full Navier-Stokes simulations. Being explicit, the code is easily vectorized to make use of current developments in parallel processing.

In addition to the "Locally Modified QUICK Scheme . . .", a refereed paper appearing in the Proceedings of the Montreal conference, the editors of the International Journal for Numerical Methods in Fluids have solicited an expanded version of this paper for a special issue of the journal devoted to about fifteen papers representing significant contributions from the Montreal meeting. This will also form the basis of an ICOMP NASA-TM. The main features of SHARP and UTOPIA were presented at an ICOMP Seminar at LeRC on July 1. Three additional papers: (1) "Simple High Accuracy Resolution Program," (2) Uniformly Third Order Polynomial Interpolation Algorithm," (3) "A Fresh Approach to Convective Flux Limiters," will be presented at the 40th Annual Meeting of the American Physical Society's Division of Fluid Dynamics in Eugene, Oregon this November. Expanded versions of 2 and 3 will also be drafted as ICOMP NASA TM's.

CHIN-SHUN LIN, Iowa State University

Numerical calculations for confined, axisymmetrical, turbulent reacting flows were studied. The k-ε turbulence model, the two-step kinetic reaction mechanism, and the Arrhenius type/eddy-breakup reaction rate were employed. Attention was paid to the local mean gas properties. The work has been presented in NASA TM 89842 (ICOMP-87-2).

Similar solutions for viscous hypersonic flow over a 3/4-power body of revolution were also studied. The results have been submitted for a NASA TM report. In this study, the effects of wall temperature, mass bleeding and body transverse curvature were investigated. The induced pressure, displacement thickness, skin friction and heat transfer in the strong interaction region were analyzed.

Numerical calculations for a hypersonic inlet flow including real gas effects are being studied. The thermodynamic and transport properties of equilibrium air is obtained from approximate curve fits. Algebraic grid generation is also contained in the PNS computer code. The results will be compared with the measurements conducted at NASA Lewis Research Center for a Mach 7.4 hypersonic inlet.

JONG-SHANG LIU, Case Western Reserve University

The objective of the study is to develop an accurate, efficient numerical analysis of the unsteady flow in a cascade subject to upstream and downstream disturbances. Three aspects of the problem are under investigation: boundary conditions, grid systems, and numerical techniques.

An existing two-dimensional, explicit, Runge-Kutta, time-marching code is used to analyze the unsteady flow through one rotor passage. The upstream and downstream boundary conditions are being modified to permit unsteady distortions because of adjacent blade rows. The profiles of the incoming wakes from the upstream stator row are specified at the upstream boundary. The use of unsteady characteristic equations and a perturbation approach at the downstream boundary is under investigation. The grid studies are being performed to determine the mesh density necessary to resolve propagating wave systems. Both the C-grid and H-grid are used for the computation. The H-grid advantage over the C-grid is that it is easy to refine the grid size in any particular region and capture the wave propagation on the upstream and downstream boundaries; however, the C-grid is smoother than the H-grid near the round leading and trailing edges of the blade. Implicit modifications to the existing explicit technique will be examined as a means of improving the computational efficiency of the code.

A paper entitled "Navier-Stokes Cascade Analysis With a Stiff k-ε Turbulence Solver" (NASA TM 100218, ICOMP 87-6, AIAA paper 88-0594) submitted to the 1988 AIAA Aerospace Sciences Meeting has been accepted for presentation.

JEFFREY P. MEISTER, University of Akron

The major focus of my work has been in developing and upgrading structural mechanics computer codes in order to increase their capabilities, ease in use, and the efficiency of their structure to minimize their memory requirements and execution time. The majority of the year was spent developing the Mechanics of Materials Model (MOMM). MOMM is a 3-D inelastic analysis methods program which characterizes the behavior of engine components by a beam network for approximate analyses during early phases of component design. Upon completion of the code by the original contractor, the user was required to devise a substitute beam network that would accurately represent the behavior of the actual component. This was an extremely inconvenient task which introduced a large degree of approximation into the analysis. I implemented a technique to internally generate an appropriate substitute beam network, thus relieving the user of this difficult task. In addition to this advancement, I also implemented a nonlinear solution algorithm, several nonlinear constitutive models, and transient analysis capabilities.

The second project I worked on this year involved a composite blade analyzer developed at Lewis Research Center called COBSTRAN. It was necessary to modify the memory allocation scheme in a more efficient manner. Previously, data was stored in numerous arrays with fixed dimensions, and the code was modified so that all data required is stored in a master array with pointers to identify the location of the various data. These modifications led to more efficient use of memory and allow the user to analyze larger problems simply by increasing the dimension of the master array.

Recently, I began working with the Engine Structures Modeling Software System (ESMOSS). ESMOSS is a software system for the construction of geometric and discrete models of engine components which can be transferred to finite element analysis programs such as NASTRAN. I have investigated ESMOSS's features and am beginning to incorporate features which will allow the user greater flexibility in modeling engine parts, such as additional types of elements that may be constructed and a topological zoom feature for mesh refinement in critical regions.

CHARLES MERKLE, Pennsylvania State University

I was a visiting researcher at ICOMP for two weeks during July. During this period most of my time was spent in becoming acquainted with the people and research programs at LeRC in the general area of CFD. Specific topical areas in which I have had interactions include techniques for computing steady and unsteady incompressible flow (W-Y Soh, J. Goodrich and H. T. Huynh);

implementation of steady and unsteady LU solvers (S-T Yu and J. S. Shuen); preconditioning techniques in hypersonic flow and flux-vector splitting (M-S Liou); two-point control difference algorithms (J. E. Lavery); acceleration procedures for computational algorithms (A. Sidi); unsteady flow computations (J. R. Scott); and multistage turbomachinery modeling (J. J. Adamczyk). I have also had the opportunity to discuss the overall CFD program with P. Sockol and to visit the experimental facilities of J. E. O'Brien and D. L. Bulzan. While here I presented an ICOMP Seminar entitled "Computation of Low Mach Number Flows."

As part of my visit I have exchanged publications with several LeRC researchers, and have made plans to exchange some special purpose computer subroutines. Finally, we have made tentative plans to continue several of these interactions after I return to Penn State, and have discussed the possibilities of return visits at a later date.

VALLORIE J. PERIDIER, Lehigh University

I am presently investigating the unsteady boundary layer due to a self-convected rectilinear vortex above a plane wall, with viscosity "turned on" at t = o. A distinguishing feature of this particular analysis is that the boundary layer equation is cast in Lagrangian dependent variables (i.e. fluid particle displacement as f(t)) rather than the traditional Eulerian formulation (i.e. local fluid velocity). The benefit of using a Lagrangian formulation is that displacements vary more slowly than velocities; consequently this analysis
has been carried out further in time than prior investigations.

The above analysis demonstrates that a narrow singular region evolves in the boundary layer. This summer at ICOMP I explored methods of capturing the boundary layer's singular evolution using an inviscid/viscous-interaction analysis.

MAJID RASHIDI, Case Western Reserve University

Active Vibration Control has become an attractive alternative to Passive Vibration Control to eliminate the undesirable vibrations of a dynamic system. In most cases, the conventional Active Vibration Control of dynamic systems (such as helicopters, flexible space telescopes, rotors, etc.) is based upon elaborate mathematical models and/or known modal characteristics of such systems. The shortcomings of the conventional active control strategies are in two general areas. First, the dynamic system is modeled with linear mathematical equation (s). Second, the modal characteristics of a physical system may change during its operation due to changes in dimension tolerances, crack initiations, wears, etc. A new active control strategy has been proposed to overcome these deficiencies.

In this new approach, the dynamic system is treated as a black-box whose modal characteristics are not included in the active control process. Instead, the FFT amplitudes of the vibration response of the system at several locations are utilized to arrange an objective function (which could behave nonlinearly) to be minimized. The decision variables are the magnitudes and phase angles of alternative force components applied to stationary points of the dynamic system.

The unbalance response of a typical rotor system was simulated on the Cray Computer System at NASA Lewis Research Center. The maximum amplitude of vibration at the center of the rotor (Disc) was determined to be 35 mils before attempting to suppress the vibration of the system via the new Spectral Optimization Strategy. After executing the new approach, the level of vibration was reduced from 35 mils to 0.5 mils.

The present results show that the large amplitudes of the unbalance response of a rotating shaft, with the disc at its center carrying the unbalance mass, can be reduced to a satisfactory level (35 mils to 0.5 mils). However, this new approach must be checked for vibration responses made up of multiple frequencies. In such cases the objective function of the Spectral Optimization will include several peaks of the FFT of the response at different frequencies.

ASHER A. RUBINSTEIN, Tulane University

An important aspect of dealing with modern materials in the aerospace industry is to be able to predict material fracture resistance on the basis of its microstructure.

The aim of this project is to identify and evaluate the parameters controlling a crack path trajectory development. This analytical study was performed with the cooperation of an experimental effort conducted at Case Western Reserve University and continuing at University of Illinois at Chicago under supervision of Professor Chudnovsky. The experimental data representing crack propagation along a curvilinear trajectory was obtained by positioning a hole in the vicinity of the crack path.

The corresponding analysis was developed partially under ICOMP support. The analysis is based on numerical simulation of the curvilinear crack path corresponding to experimental geometry and stress state. A numerical approach for representing a curvilinear crack is based on the representation of the crack by an array of dislocations along the curvilinear path. This formulation leads to a singular integral equation of the first kind with a Cauchy type singularity. The formulation is done in terms of complex variables and with an unknown complex valued dislocation density distribution function. The numerical scheme is based on Gaus-Chebyshev quadrature. This collocation scheme involves nodal distribution at the roots of Chebyshev polynomials of the first and second kind. For the case of curvilinear crack path leads it becomes necessary to solve a nonlinear equation at each point in order to locate the exact position of the nodes. This numerical solution was carried out on the Cray X-MP supercomputer at the NASA Lewis Research Center.

Experimentally obtained crack trajectories were represented by polynomial curves using a least square approximation. Thus using numerical simulation of the fracture process, one was able to observe variation of the important fracture parameters along the crack path trajectories as the crack grows and the variations of these parameters in the event of possible crack path deviations. Results demonstrate important features of the crack path formation.

NESRIN SARIGUL, Ohio State University

A family of hermitian boundary traction elements originally developed by the author were updated to utilize explicit integration in the stiffness and mass matrices computations including membrane and bending behavior.

It is well known that finite elements with reduced and/or selective integration are adapted in various schemes to overcome the overstiff predictions of displacements in finite element analysis of structures. Membrane locking and shear locking are also well discussed in this context. It is anticipated that the exact integration scheme developed by the author will alleviate these problems.

At present, an exact integration scheme is incorporated in a computer code for blending finite elements. These elements will be utilized to obtain the frequency response of a turbine blade. It is anticipated that this research will bring about a new technique in dealing with finite element formations. Once the research is completed, the results will be published in Numerical Methods related journals.

The computer codes developed are in Fortran and available at VAX series computers. It may be noted that the parallel processors are good candidates for practical applications of this development.

STEVEN SCHAFFER, New Mexico Institute of Technology

A computer code called MG01 was developed by Rod Chima and myself to solve the steady two dimensional Euler and Navier-Stokes equations for flow through a channel with a circular bump on a wall. The algorithm on which the code is based uses the full approximation multigrid method with a semi-explicit Runge-Kutta scheme as a driver. It was found that the scheme slowed down considerably for grids with high aspect ratios. It was hypothesized that the slow convergence rate was due to the pointwise nature of the Runge-Kutta scheme. Attempts were made to introduce a more directionally biased implicit structure to the semi-implicit Runge-Kutta scheme. It was found that the Runge-Kutta scheme was not amenable to such implicit treatment and difficulties in the development of a more robust scheme were further compounded by the fact that our multigrid Runge-Kutta scheme is rather sensitive to the choice of parameters.

In light of these shortcomings, it was tempting to compare this algorithm with a similar multigrid algorithm that used a more implicit driver. To this end, an incomplete LU decomposition scheme based on the flux vector splitting of Jameson and Turkel was added to the code. A line Gauss-Seidel scheme is also being added to the code. At this point, both algorithms are still under development. Comparison of these three algorithms should enhance understanding of multigrid methods for mixed elliptic-hyperbolic and hyperbolic systems of differential equations and lead to more robust and faster steady solvers.

A second research project was proposed in which the steady codes could be used to calculate unsteady flows. The basic idea here is to start with an implicit scheme for the unsteady equations and use the steady solver to obtain an approximate solution of the implicit equations at each time step. This can be accomplished with only slight modifications to the existing steady code.

To illustrate, consider the two dimensional unsteady Euler equations,

$$Q_t + E_x[Q] + F_y[Q] = 0 \tag{1}$$

where

$$Q = [\rho, \rho u, \rho v, e]^T$$

is the vector of conserved dependent variables and

$$E = [\rho u, \rho u^2 + P, \rho uv, [e + P]u]^T$$

$$E = (\rho v, \rho uv, \rho v^2 + P, [e + P]v]^T$$

are the flux vectors in the x and y directions, respectively. Using backward Euler, say, to march in time, we obtain at each time step the equations

$$Q^h[t + \Delta t] + \Delta t\, R^h(Q^h[t + \Delta t]) = Q^h[t] \tag{2}$$

where $Q^h[t]$ denotes the discrete approximation to $Q[t]$ at time t and R^h denotes the central difference operator approximating the flux terms and artificial viscosity. Let $M^h[W^h] = W^h + \Delta t R^h[W^h]$ and $G^h = Q^h[t]$ so that [2] can be written as

$$M^h[W^h] + G^h \tag{3}$$

where $W^h = Q^h\,[t + Wt]$. With slight modification, the steady solver designed to solve $R^h[Q^h] = 0$, can be used to solve [3]. For instance, the pseudo-time equations

$$W^h t + M^h[W^h] = G^h \tag{4}$$

$$W^h|_{\tau = o} = Q^h[t]$$

could be driven to steady state

$$\lim_{\tau \to o} W^h[\tau] = Q^h[t + \Delta t]$$

using the Runge-Kutta or LU multigrid schemes.

Advantages of this scheme are: [1] the steady solvers exist and can be easily modified to solve (4); [2] as the efficiency of the steady solvers is improved, so should the corresponding unsteady schemes; [3] the accuracy of the time calculations can be controlled by the number of pseudo-time steps taken at each real time step and; [4] the amount of work to obtain a solution at each time step should be comparable to the approximate solution of (2) using a single grid method.

JAMES SCOTT, University of Dayton

During my participation in ICOMP, I addressed issues associated with the accuracy of numerical solutions of the governing equations of fluid dynamics.

The primary emphasis of this effort was directed toward assessing the nature of the truncation error of various numerical schemes. The accuracy of a numerical scheme is analyzed by examining the leading term in the truncation error and the gain factor which is determined from stability analysis. If the leading term in the truncation error contains an even derivative, the error is dissipative while an odd derivative indicates that the error is dispersive. Stability analysis yields the gain factor in which the amplitude gives the magnitude of the dissipative error and the phase gives the magnitude of the dispersive error.

Of particular interest is the MacCormack explicit finite difference algorithm which is a two-step Lax-Wendroff algorithm. When this algorithm is used to solve the time-dependent Navier-Stokes equations it is second order accurate in both time and space. This scheme is often said to be dissipative. However, examination of the truncation error indicates that when the predictor and correl. steps are combined the leading term in the truncation error contains a third derivative and hence is dispersive rather than dissipative. Careful examination of previous investigations revealed that the numerical dissipation introduced in MacCormack's scheme (and Lax-Wendroff schemes in general) should be of concern only in regions of high pressure gradients such as shocks or rapid expansions. In general, the numerical dissipation is on the order of 100 to 1000 times smaller than the molecular viscosity. In addition, a recent comparison of the dissipative properties of eight numerical algorithms widely used for solving the Euler and Navier-Stokes equations indicated that MacCormack scheme is significantly better except in the vicinity of shock waves.

Other aspects of numerical accuracy were also investigated. These included the modified equation approach, higher order schemes and geometric conservation law considerations. While preliminary analyses indicate that improved accuracy can be achieved with some of these approaches, additional study is required to determine whether the results will be any better than those obtained by simple grid refinement.

In addition to numerical accuracy, I reviewed the use of turbulence models in unsteady Navier-Stokes computations. This is in keeping with the examination of the dissipative nature of specific numerical schemes since the turbulence model itself is an additional dissipative mechanism which is introduced into the computation. In fact turbulence models will completely damp unsteady features of a flow. Hence, techniques other than the classic approaches to turbulence modeling are required for unsteady flow. Review of these procedures and analysis of large scale time-dependent flow features (which can generally be regarded as deterministic) have stimulated ideas which could lead to alternative methods of accounting for turbulence in unsteady flow. These ideas will require further development in order to assess their feasibility.

AVRAM SIDI, Technion - Israel Institute of Technology

In the two summers (a total of four months) that I worked at ICOMP I explored possible modes of application of extrapolation methods for vector sequences to computational fluid mechanics. The vector sequences of utmost interest are those that arise from finite difference solution of partial differential equations in general, and Euler and Navier-Stokes equations in particular. The extrapolation methods with which I had a substantial amount of experience are the minimal polynomial extrapolation and the reduced rank extrapolation.

Most of the theoretical research on vector extrapolation methods has been carried out at NASA Lewis Research Center, Cleveland, Ohio and Technion-Israel Institute of Technology, Haifa, Israel, in the past five years. Results pertaining to the very crucial aspects, such as convergence and stability, have been very useful in developing strategies of practical value in the applications that were done subsequently.

So far the above mentioned methods have been successfully used in conjunction with two computer codes that were written for a subsonic compressible duct flow problem and two others that were written for a high speed transonic propeller problem. In all these applications substantial savings in number of iterations were achieved.

Both of the extrapolation methods above can be implemented in parallel with the code that is performing the iterations and producing the relevant vector sequence. This will reduce the overhead resulting from the use of extrapolation methods significantly. There have been some contacts here at Lewis that will hopefully result in the design of a parallel processor for extrapolation methods.

ELI TURKEL, Tel Aviv University

In joint work with A. Solomonoff we studied the properties of spectral approximations. Various functions were approximated by a global collocation method using both Chebyshev nodes and equally spaced nodes. It was observed that the properties of these methods differed from that of local methods. As an example, when a large gradient exists in the function, the global approximation is more accurate when the large gradient is near the boundary. This is true for both sets of nodal points and so the increased resolution is not due to more points in the boundary region. A paper describing these results and extension to partial differential equations has been submitted to the Journal of Computational Physics.

An effort was begun with R. Chima of NASA Lewis to investigate the multigrid scheme and compare it with the multiple level approach of R. Ni. Two central difference Runge-Kutta codes were used, one based on a finite volume approach and one based on a finite difference approach. These codes were then applied to both inviscid and viscous problems for flow about an airfoil and flow in a duct. The standard full multigrid algorithm was found to be more efficient than the Ni approach. Some theoretical work supporting this conclusion was also found. An AIAA paper was presented at the 1987 Aerospace Sciences Meeting in Reno, Nevada.

LEON van DOMMELEN, Florida State University

A number of applications in computational fluid dynamics allow the flow to be described using a relatively small number of discrete elements: examples are panels and vortices. Description using such elements can reduce storage requirements, simplify algorithms, and allow complex configurations and infinite domains to be described. In some applications, the elements are of particular interest by themselves: vortices in liquid Helium or stars in galaxies. In other cases they are used to describe sharp distributions, such as vortices in separated flow at high Reynolds number.

Unfortunately, the computational effort tends to increase rapidly with the number of elements, since each element is affected by all other elements individually. In order for these methods to remain competitive for increasing number of elements, it is desirable to separate the elements closely together from the elements further apart. Approximations may then be made for the elements not in the immediate vicinity of each other. Such a procedure was developed by van Dommelen & Rundensteiner [1] for two-dimensional flow and led to sizable reductions in computational time.

The objective of this study was to examine possible extensions to three dimensions. As example, the gravitational forces introduced by point masses were determined numerically. This problem is closely related to the problem of the velocity introduced by vorticity, but somewhat simpler since mass is scalar while vorticity is a vector.

In the procedure developed, the domain is recursively subdivided into cubes until the number of masses inside each cube is less than a pre-determined maximum. The cubes are numbered using a generalization of the procedure [1] in which the digits of the three binary cube coordinates are combined into a single base-3 number.

Important differences between the two-dimensional and three-dimensional procedures are: portable Fortran (66 or 77) rather than Fortran 200, yet written to vectorize readily on Cray and CDC vector processors; changed storage of the cube numbers to allow much more subdivisions; reduction in the storage required for each cube by about a factor 7.

Since asymptotic approximations of high order are much less effective in three dimensions, the scheme was modified to restrict approximations to larger distances; to compensate the present scheme allows much more subdivisions of the domain.

The program is operational and some first tests have been run on the NASA Lewis VAX cluster. Some inconsistencies in charged CPU time were noted, possibly due to paging, but they should not sizably affect the conclusions.

In general it was found that the overhead occurred in generating and addressing the cubes was negligible; the savings in computational time were directly proportional to the number of terms that could be replaced by asymptotic approximations.

For the representative case of 2000 masses arranged on a circle, the present algorithm reduces the computational time by a factor 3.5. The algorithm does not lead to reduced accuracy: the original sum is poorly conditioned and its 2% relative error dominates any errors in the asymptotic approximations. For 4000 masses the savings were somewhat limited by the desire to keep the storage equivalent to the original algorithm; a factor 4.5 in savings was achieved.

In order to study the errors introduced by the algorithm, these cases were repeated in 64 bit precision. For 2000 masses the savings were here a factor 3.7, for 4000 a factor 6.2 and for 8000 a factor 10.9. The relative error in the magnitude of the force was found to be negligible, while the error in direction varied from 0.09% for 2000 masses to 0.27% for 8000.

For masses arranged on a circular disk, it was found that the applicability of the asymptotic approximations was more limited. Even at 8000 masses, the savings were only 25%. This could possibly be improved by higher order asymptotic approximations or accepting less than 5 digits accuracy.

While there can be no doubt that the three dimensional problem is much more difficult than in two dimensions, the preliminary results obtained in this study are encouraging. They show clearly that sizable reductions in effort can in fact be achieved even for only a few thousand points.

REFERENCE

1. van Dommelen, L. L. & Rundensteiner, E. A. "Fast Solution of the Two-Dimensional Poisson Equation with Pointwise Forcing," submitted to the J. Comp. Phys.

BRAM van LEER, University of Michigan

Several aspects of flux-splitting techniques were studied in collaboration with M.-S. Liou. In the first place, known formulas for flux-vector and flux-difference splitting were extended to incorporate real-gas effects. The motivation of this work lies in the current interest in high-speed high-temperature flows. A summary of this work was submitted to the First National Fluid Dynamics Congress, Cincinnati, Ohio, July 1988.

Another research subject regarding flux-splitting methods is to come up with the best strategy of driving flux-split residuals to zero in steady-state calculations. Various combinations of flux-split and flux-difference-split residuals with implicit operators were analyzed. One preliminary conclusion is that the Steger-Warming flux-vector splitting is the most robust choice for the implicit operator. This work will be reported in a paper that has been accepted for the 26th Aerospace Sciences Meeting in Reno, January 1988.

J.D.A. WALKER, Lehigh University

My first effort was concerned with unsteady boundary-layer separation. Unsteady flow separation occurs in a wide variety of physical situations such as the flows that are commonly encountered in turbomachinery and over a pitching airfoil surface. In most situations, the separation process initiates in a viscous boundary layer near the surface in the form of the appearance of a closed recirculating eddy; depending on the particular flow environment, the eddy may or may not be attached to the surface. At this stage of development, the flow field may still be considered to be double structured, consisting of an outer inviscid flow over the surface and an (as yet) thin viscous flow adjacent to the surface. With the passage of time, the boundary layer is then observed to enter a phase of substantial and accelerating growth in a direction normal to the surface at streamwise locations close to the eddy. The process culminates in a strong inviscid-viscous interaction in which an eruption of the boundary-layer flow out into the inviscid region occurs. This boundary-layer eruption may take several forms, depending on the local external flow conditions and the geometry involved, but often results in the ejection of a secondary vortex from the boundary-layer flow near the surface. For example, in the case of leading-edge separation bubbles or turbine blades or pitching airfoils, the bubble is often observed to erupt and shed into the outer flow in a variety of circumstances.

The present research is associated with the development of numerical algorithms which will permit the calculation of an evolving unsteady flow near a surface through into interaction with the outer flow at high Reynolds numbers. Most conventional procedures (as well as full Navier-Stokes methods) encounter severe numerical difficulty in continuing a numerical integration once a boundary layer eruption begins to take place. At high Reynolds numbers, such eruptions occur over a band of very narrow streamwise extent at locations which are *a priori* unknown. The present algorithms use a Lagrangian description to track the evolution of the erupting viscous flow. Interacting boundary layer theory is used to couple the calculations with the outer flow. The results obtained are very encouraging and suggest that the present methods may have wide applicability.

My second activity was concerned with embedded function methods. In conventional numerical methods for the prediction of turbulent flows near walls, substantial numbers of mesh points and computational effort is required to adequately resolve the intense velocity and temperature profile variations that occur in the near wall region. In this research, algorithms are being developed wherein an outer numerical solution is matched to a set of embedded analytic profile functions which describe the turbulent mean profiles near the wall. These wall-layer profiles have been developed through consideration of the coherent structure of the time-dependent near-wall flow. Skin friction and heat transfer coefficients are obtained through asymptotic matching conditions which are the mathematical statements that the outer region numerical solution should join smoothly onto the analytical wall layer profiles. Current studies indicate that only half the mesh points (as opposed to conventional methods) are required in this approach; the algorithms being developed have proved to be very efficient and accurate for two dimensional turbulent flows and are currently being considered for three-dimensional problems. It is anticipated that this methodology will be extended to full Navier-Stokes methods to substantially reduce the number of mesh points near solid walls and to improve computational efficiency.

CHOWEN CHOU WEY, Georgia Institute of Technology

To study the variable density effects in decaying grid turbulence, Direct Numerical Simulation, which is to directly solve the time-dependent Navier-Stokes equations, is employed. The turbulent mixing layer is taken to be statistically homogeneous in the lateral (y) and flow (x) directions. That is, the statistical properties of the flow depend only on the transverse coordinate (z) and time. This enables us to compute temporally developing mixing layers instead of the spatially developing ones usually studied in the laboratory.

Pseudo-spectral numerical methods are used to solve the governing equations of motion. The variables are expanded in Fourier Series. Periodic boundary conditions are used in the directions of homogeneity (x,y) and free-slip conditions in the transverse (z) direction.

Various initial conditions, such as fundamental and/or subharmonic forcing, intensity of the turbulence, and Reynolds Number, are tested. Series of numerical simulations are also performed to examine the impact of increasing the degree of density stratification, starting with the constant density mixing layer, in a "simple" flow field. The comparison between constant and variable mixing layer reveals some interesting phenomena, but the exact explanation is still under study.

DAVID WHITFIELD, Mississippi State University

During the two-weeks at ICOMP from August 2-14, 1987, attention was focused on the computation of flow associated with two rotating machinery problems. The first was the flow through NASA's proposed supersonic through-flow compressor. This involved the three-dimensional unsteady solution of the Euler equations for a rotor-stator combination. Very impressive color graphics of this solution were produced by NASA personnel, and the necessary files were constructed for the time animation of the unsteady solution.

The second problem also required the solution of the three-dimensional unsteady Euler equations, this time for the Hamilton-Standard SR-7 propfan. Attention here was given to the development of a computational method to support an experimental test program recently conducted in France. Because the test program could not handle the entire eight blades only two blades were used in the experiment. Therefore, one of the objects of the computational effort is to investigate cascade effects to assess how the experimental data might be extrapolated to the true eight-blade configuration.

YAU SHU WONG, University of Alberta, Edmonton

Many practical problems in science and engineering require the solution of large systems of equations. The conjugate gradient (CG) method used in conjunction with a suitable preconditioning technique has been widely accepted as one of the most efficient iterative procedures. The basic CG algorithm can be efficiently implemented on a vector computer such as the CDC CYBER 205. The use of a matrix polynomial as a preconditioner has been proposed, and it is also suitable for coding on a vector computer. However, the amount of computational work due to the polynomial preconditioning is larger than the basic CG algorithm itself. This leads us to investigate an alternative choice for a more efficient preconditioner. A new technique, namely, the approximate polynomial preconditioning, has been shown to provide a significant improvement over the traditional polynomial preconditioning. The basic idea is that the preconditioning polynomial is derived from another coefficient matrix that contains far fewer nonzero elements than the original matrix operator. The detail of this technique and the numerical results for solving biharmonic equations will be presented in the ICOMP report.

Balarini, Roberto (CWRU) and Plesha, Michael E. (ICOMP): "The Effects of Crack Surface Friction and Roughness on Crack Tip Stress Fields." ICOMP Report No. 87-1, NASA TM 88976, February 1987, 17 pages.

A model is presented which can be used to incorporate the effects of friction and tortuosity along crack surfaces through a constitutive law applied to the interface between opposing crack surfaces. The problem of a crack with a saw-tooth surface in an infinite medium subjected to a far-field shear stress is solved and the ratios of Mode-I stress intensity to Mode-II stress intensity are calculated for various coefficients of friction and material properties. The results show that tortuosity and friction lead to an increase in fracture loads and alter the direction of crack propagation.

Lin, Chin-Shun (ICOMP): "Numerical Calculations of Turbulent Reacting Flow in a Gas-Turbine Combustor." ICOMP Report No. 87-2, April 1987, 20 pages.

A numerical study for confined, axisymmetrical, turbulent diffusion flames is presented. Local mean gas properties are predicted by solving the appropriate conservation equations in the finite-difference form with the corresponding boundary conditions. The k-ε two-equation turbulence model is employed to describe the turbulent nature of the flow. A two-step kinetic model is assumed to govern the reaction mechanism. The finite reaction rate is the smaller of an Arrhenius type of reaction rate and a modified version of eddy-breakup model. Reasonable agreement is observed between calculations and measurements, but to obtain better agreement, more work is needed on improvements of the above mathematical models. However, the present numerical study offers an improvement in the analysis and design of the gas turbine combustors.

Givi, Peyman (ICOMP) and Kosaly, George (University of Washington): "On the Coalescence-Dispersion Modeling of Turbulent Molecular Mixing," ICOMP Report No. 87-3, NASA TM 89910, July 1987, 24 pages.

The general coalescence-dispersion (C/D) closure provides phenomeno- logical modeling of turbulent molecular mixing. The models of Curl, and Dopazo and O'Brien appear as two limiting C/D models that "bracket" the range of results one can obtain by various models. This finding is used to investigate the sensitivity of the results to the choice of the model. Inert scalar mixing is found to be less model-sensitive than mixing accompanied by chemical reaction. Infinitely fast chemistry approximation is used to relate the C/D approach to Toor's earlier results. Pure mixing and infinite rate chemistry calculations are compared to study further a recent result of Hsieh and O'Brien who found that higher concentration moments are not sensitive to chemistry.

Ghoniem, Ahmed F. (MIT) and Givi, Peyman (ICOMP): "Vortex-Scalar Element Calculations of a Diffusion Flame Stabilized on a Plane Mixing Layer," ICOMP Report No. 87-4, NASA TM 100133, August 1987, 25 pages.

The vortex-scalar element method, a scheme which utilizes vortex elements to discretize the region of high vorticity and scalar elements to represent species or temperature fields, is utilized in the numerical simulations of a two-dimensional reacting mixing layer. Computations are performed for a diffusion flame at high Reynolds and Peclet numbers without resorting to turbulence models. In the non-reacting flow, the mean and fluctuation profiles of a conserved scalar show good agreement with experimental measurements. Results for the reacting flow indicate that for temperature-independent kinetics, the chemical reaction begins immediately downstream of the splitter plate where mixing starts. Results for the reacting flow with Arrhenius kinetics show an ignition delay, which depends on the reactants temperature, before significant chemical reaction occurs. Harmonic forcing changes the structure of the layer, and concomitantly the rates of mixing and reaction, in accordance with experimental results. Strong stretch within the braids in the nonequilibrium kinetics case causes local flame quenching due to the temperature drop associated with the large convective fluxes.

Wong, Yau Shu (ICOMP) and Jiang, Hong (University of Alberta): "Approximate Polynomial Preconditioning Applied to Biharmonic Equations on Vector Supercomputers," ICOMP Report No. 87-5, NASA TM 100217, October 1987, 19 pages.

Applying a finite difference approximation to a biharmonic equation results in a very ill-conditioned system of equations. This paper examines the conjugate gradient method used in conjunction with the generalized and approximate polynomial preconditionings for solving such linear systems. An approximate polynomial preconditioning is introduced, and is shown to be more efficient than the generalized polynomial preconditionings. This new technique provides a simple but effective preconditioning polynomial, which is based on another coefficient matrix rather than the original matrix operator as commonly used.

Liu, Jong-Shang (ICOMP); Sockol, Peter M. (NASA Lewis); and Prahl, Joseph M. (CWRU): "Navier-Stokes Cascade Analysis with a Stiff k-ε Turbulence Solver," ICOMP Report No. 87-6, NASA TM 100218, prepared for the AIAA 26th Aerospace Sciences Meeting, Reno, Nevada, January 11-14, 1988, 14 pages.

The two-dimensional, compressible, thin-layer Navier-Stokes equations with the Baldwin-Lomax turbulence model and the k-ε model are solved numerically to simulate the flow through a cascade. The governing equations are solved for the entire flow domain, without the boundary layer assumptions. The stiffness of the k-ε equations is discussed. A semi-implicit, Runge-Kutta, time-marching scheme is developed to solve the k-ε equations. The impact of the k-ε solver on the explicit Runge-Kutta Navier-Stokes solver is discussed. Numerical solutions are presented for two-dimensional turbulent flow over a flat plate and a double circular arc cascade and compared with experimental data.

Lin, Chin-Shun (ICOMP): "Similar Solutions for Viscous Hypersonic Flow Over a Slender Three-Fourths-Power Body of Resolution." ICOMP Report No. 87-7, NASA TM 100205, 22 pages.

For hypersonic flow with a shock wave, there is a similar solution consistent throughout the viscous and inviscid layers along a very slender three-fourths-power body of revolution. The strong pressure interaction problem can then be treated by the method of similarity. In the present study, numerical calculations are performed in the viscous region with the edge pressure distribution known from the inviscid similar solutions. The compressible laminar boundary-layer equations are transformed into a system of ordinary differential equations. The resulting two-point boundary value problem is then solved by the Runge-Kutta method with a modified Newton's method for the corresponding boundary conditions. The effects of wall temperature, mass bleeding, and body transverse curvature are investigated. The induced pressure, displacement thickness, skin friction, and heat transfer due to the previously mentioned parameters are estimated and analyzed.

The ICOMP Steering Committee: "Institute for Computational Mechanics in Propulsion (ICOMP), First Year Summary," ICOMP Report No. 87-8, 14 pages.

The Institute of Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. The Institute began operation in 1985. This report describes the events leading to its formation, its organization and method of operation, and the technical activities of the first year.

Leonard, B. P. (ICOMP): "SHARP Simulation of Discontinuities in Highly Convective Steady Flow," ICOMP Report No. 87-9, 34 pages.

For steady multidimensional convection, the QUICK scheme has several attractive properties. However, for highly convective simulation of step profiles, QUICK produces unphysical overshoots and a few oscillations, and this may cause serious problems in nonlinear flows. Fortunately, it is possible to modify the convective flux by writing the "normalized" convected control-volume face value as a function of the normalized adjacent upstream node value, developing criteria for monotonic resolution without sacrificing formal accuracy. This results in a nonlinear functional relationship between the normalized variables, whereas standard methods are all linear in this sense. The resulting Simple High Accuracy Resolution Program (SHARP) can be applied to steady multidimensional flows containing thin shear or mixing layers, shock waves, and other frontal phenomena. This represents a significant advance in modeling highly convective flows of engineering and geophysical importance. SHARP is based on an explicit, conservative, control-volume flux formulation, equally applicable to one-, two-, or three-dimensional elliptic, parabolic, hyperbolic, or mixed-flow regimes. Results are given for the bench-mark purely convective oblique-step test. The monotonic SHARP solutions are compared with the diffusive first-order results and the nonmonotonic predictions of second- and third-order upwinding.

DR. EDWARD A. BOGUCZ, Syracuse University: "Calculations of Unsteady Thermal Boundary Layers and Turbulent Wake Flows"

Unsteady Boundary Layer Heat Transfer Due to the Motion of Free-Stream vortices: The effects of free-stream vortices on the flow and heat transfer in a surface boundary layer are considered for the case of a circular cylinder, which is exposed to an approaching symmetric vortex pair that is imbedded in an otherwise uniform flow. A numerical solution is formulated for the hydrodynamic and thermal boundary layers at the stagnation point. Calculated results for several vortex flow situations are presented. Recent LeRC measurements of unsteady heat transfer on a simulated stator blade, which is subjected to passing wakes that contain discrete vortices, are discussed in terms of the present results.

Turbulent Wake Calculations: The modeling and calculation of turbulent wake flows is discussed in view of recent analytical and experimental work. Numerical solutions of the complete boundary layer equations for the symmetric wake of a thin flat plate are shown to support a recent analytical description of the near wake region, which is based on an asymptotic analysis of the governing equations. Calculations of the wake development downstream of the near wake region are presented. The behavior of eddy viscosity models required for accurate predictions is discussed.

DR. LOLA BOYCE, The University of Texas at San Antonio: "Improving the Reliability of Space Propulsion Systems: Development of Computational Methods for Material Strength"

Material strength degradation is quantified by damage indicators such as crack length, fatigue strength or fiber degradation that change monotonically over time. Experimental data that relate damage and time have been represented as deterministic mathematical models using theories such as fracture mechanics and micromechanics constitutive theory. Models predict the "time to reach a certain material damage level." Thus space propulsion system component lifetimes may be estimated.

Experience has shown that under fatigue loading such estimates only roughly correlate with observed lifetimes, since fatigue is essentially a random phenomenon. This talk discusses the probabilistic treatment of these material strength degradation models. Probabilistic analysis methods such as simulation and moment transfer methods are discussed. Computer programs utilizing simulation, and both parametric and non-parametric methods of estimating the "random time to reach a random material damage level" are discussed. These probabilistic methods increase the information available upon which to estimate the lifetime of space propulsion system components.

DR. FRED S. H. CHANG, Cleveland State University: "Shock Capturing Schemes for Conservation Laws"

Total variation may be used as a measure of oscillation. One way to design non-oscillatory shock capturing schemes is to require the total variation of the numerical solution be non-increasing. This leads to the construction of TVD (total variation diminishing) schemes. Allowing the total variation to possibly increase, but only by an amount on the level of local truncation errors, and using a certain interpolation procedure lead to uniform-order essentially non-oscillatory (ISNO or ENO) schemes. The construction of ENO schemes is discussed and computational results are presented.

DR. TIAO J. CHANG, Ohio University: "Stochastic Approach for Studying Occurrences of Rare Events"

Natural phenomena may not be governed by deterministic equations. The governing equations may not be solved deterministically. The stochastic approach could be an alternative. A stochastic process, namely non-homogeneous Poisson process, is developed to model occurrences of the rare events. The formulation of the intensity function that characterizes the severity of rare events is discussed.

DR. STEPHEN S. COWLEY, Imperial College of Science & Technology, London, England: "Numerical Solutions of the Unsteady Triple-Deck Equations"

A brief introduction to "High"-Reynolds-number triple-deck theory for laminar boundary-layer flows will be given. For steady flows triple-deck theory explains boundary-layer separation without the occurrence of a Goldstein singularity. For unsteady flows the triple-deck formulation is not as successful in desingularizing the boundary-layer equations. Calculations (in progress) to identify singular behavior will be described.

DR. W. N. DAWES, Whittle Laboratory, University of Cambridge, Cambridge, England: "Computation of 3D Viscous Compressible Flow in Axial and Radial Flow Turbomachinery"

A simple, semi-implicit multigrid time marching code is described, written for application to any turbomachinery geometry. Applications are illustrated for axial flow turbines, axial compressor (with clearance flow), radial inflow turbine, centrifugal compressor and axial-radial steam turbine exhaust diffuser.

DR. JOHN D. DENTON, Whittle Laboratory, University of Cambridge, Cambridge, England: "Can we Compute Losses in Turbomachines?"

The mechanisms of loss generation in turbomachines will be described and the difficulty of computing the loss quantitatively will be highlighted. It will be suggested that many of the viscous flow phenomena in a turbomachine are scarcely understood, let alone calculable. In this situation it is premature to expect quantitative results from viscous flow calculations, and it is suggested that "semi viscous" calculations with considerable empirical input are preferable. The talk will be illustrated by results from viscous and semi viscous two-dimensional and 3D calculations.

DR. GEORGE S. DULIKRAVICH, Pennsylvania State University: "Interdisciplinary Research in Computational Mechanics: Ideas and Results"

For each of the following topics, we will focus on a discussion of results giving only a brief review of the underlying concepts: 1. Computational grid generation using optimization, 2. Acceleration of iterative algorithms for non-linear problems, 3. Physically dissipative full potential equation for unsteady compressible flows, 4. Numerical dissipation based on physical dissipation, and 5. Inverse problems (design) in fluid mechanics and heat transfer.

DR. PETER R. EISEMAN, Columbia University: "A Control Point Form of Algebraic Grid Generation"

Local control points are established within the context of algebraic grid generation. The method of generation is based upon a multi-directional assembly of multisurface transformations that incorporates the best features of tensor product and Boolean sum constructions. Upon assembly, the resultant capability is the capacity to conform precisely to prescribed boundaries while being able to manipulate the grid through a sparse net of control points. An interacting code has been developed in two-dimensions on the IRIS 2500. An on-line demonstration of that code will be given as a part of this seminar.

DR. PETER EISEMAN, Columbia University: "A Tutorial on Algebraic Grid Generation in Three Sessions"

The topic of algebraic grid generation will be examined in a relaxed and informal manner so that the audience can actively participate in the discussion. The aim is to deepen the understanding of the subject rather than to cover a large amount of material in a short time period. In an overall progression, the discussion will start with basic invariate constructions and evolve towards control point formulations. Such a formulation is already on the NASA Lewis IRIS 3030 and was briefly described in a previous lecture.

DR. MICHAEL R. FOSTER, Ohio State University: "On the Stability of Longs' Vortex with Large Axial Momentum"

A family of long, slender vortices with conical similarity found by Long (JFM 11, 1962) has some members whose axial velocity profiles have off-axis peaks, and flow reversal near the vortex center. These profiles may have some relevance for breakdown of certain kinds of vortices. Asymptotic methods show that at large axial momentum flux, the vortex has a ring-jet structure, whose stability is dominated by the stability of the Bickley jet. However, for longer axial-wave-length disturbances, the angular momentum becomes important, and the instability becomes centrifugal. The theory contains some long-wave neutral modes. Comparison with numerical computation of the instabilities, which is excellent for the "sinuous" mode but poor for the "varicose" mode is discussed.

DR. DAVID GOTTLIEB, Tel Aviv University: "Multiple Steady States for Characteristic Initial Value Problems"

The time dependent, isentropic, quasi-one-dimensional equations of gas dynamics and other model equations are considered under the constraint of characteristic boundary conditions. Analysis of the time evolution shows how different initial data may lead to different steady states and how seemingly anamolous behavior of the solution may be resolved. Numerical experimentation using time consistent explicit algorithms verifies the conclusions of the analysis. The use of implicit schemes with very large time steps leads to erroneous results.

PROFESSOR BORIS GRANOVSKY, Case Western Reserve University: "Admissible and Optimal Procedures in Simulation by the Monte Carlo Method"

Monte Carlo method as a probabilistic method for solving linear integral equations. The rate of convergence of a general M.- K. procedure. The concept of admissibility of a M.- K. procedure and criterion for an efficient simulation scheme. Examples of admissible and optimal Monte Carlo procedures.

DR. MURLI GUPTA, George Washington University: "Spectrum Transformation Techniques for Divergent Iterations"

A number of iteration schemes diverge when the spectral radius of iteration matrix is larger than one. If the eigenvalue spectrum of the iteration matrix lies entirely within $S = [Z: | \text{REAL}(Z) | < 1]$, it can be enveloped by an ellipse lying within S, and the divergent iterations can be transformed into a convergent one. When the spectrum of the iteration matrix lies outside S, a matrix transformation can be used to reduce this problem to one whose spectrum lies in S. A code has been developed that locates the eigenvalue spectrum and automatically obtains the parameters of the transformations. Some recent results are presented. Test problems include convection - diffusion equations and the biharmonic equation.

DR. THOMAS HAGSTROM, State University of New York at Stony Brook: "Numerical Boundary Conditions and the Computation of Spatial Evolution"

We discuss the problem of computing appropriate boundary conditions at artificial boundaries for the simulation of nonlinear wave propagation. By means of a simple example, we show that the choice of such conditions can be crucial for long time calculations. We propose conditions appropriate for the study of the spatial development of perturbations to parallel, incompressible flows.

DR. S. I. HARIHARAN, University of Akron: "Accurate Boundary Conditions for Exterior Problems in Gas Dynamics"

The numerical solution of exterior problems is typically accomplished by introducing an artificial, far field boundary and solving the equations on a truncated domain. For hyperbolic systems, boundary conditions at this boundary are often derived by imposing a principle of no reflection. However, waves with spherical symmetry in gas dynamics satisfy equations where incoming and outgoing Riemann variables are coupled. This suggests that 'natural' reflections may be important. We propose a reflecting boundary condition based on an asymptotic solution of the far field equations. We obtain nonlinear energy estimates for the truncated problem and present numerical experiments to validate our theory.

DR. KAROL Z. KORCZAK, Case Western Reserve University: "Advanced Numerical Methods for Complex Flow Problems: Chebyshev and Legendre Spectral Elements"

The isoparametric spectral element method, a combination of finite elements and spectral formulations, has experienced several reformulations since its original development in 1984-85. The modifications have improved the performance of the algorithms substantially and have been directed towards commercialization of the method. In this presentation, the latest advances in the development of the algorithm based on Chebyshev and Legendre spectral expansions will be presented. The areas of discussion will include mesh generation techniques, mathematical formulation of the equations in space and time, implicit and explicit time integration, incorporation of moving boundaries and others. Description of current areas of applications and research efforts will conclude the presentation.

DR. SHIH-TUEN LEE, National Taiwan University: "A Model of Turbulent Diffusion Flame Adjacent to a Solid Fuel"

A model is developed for turbulent diffusion flame adjacent to a solid fuel. Special attention is given to the low-Reynolds number zone near the solid fuel, the interaction between the turbulence and the chemical kinetics, and soot formation and oxidation. Some preliminary results are also presented.

DR. B. P. LEONARD, University of Akron: "Modelling Highly Convective Flows"

<u>Wiggle-Free Steady (3D) Convection with Discontinuities, using SHARP--the General-Purpose All-Natural Solver (No Artificial Additives)</u>. SHARP is a Simple High Accuracy Resolution Program designed to simulate steady convection involving near discontinuities (shear and boundary layers, shock waves, etc.) without generating wiggles or overshoots. SHARP is inherently monotonic without resorting to artificial additives or complicated flux limiters. It is based on a cost-effective modification of QUICK using an Exponential Upwinding or Linear Extrapolation Refinement, but only in critical zones--QUICK is used throughout the bulk of the flow domain.

<u>Explicit Unsteady (3D) Convection-Diffusion Schemes</u>. These are control-volume (finite-domain) conservative flux formulations based on the pure-convection equation: $\phi(\underline{x}, t) = \phi(\underline{x}-\underline{v}\Delta t, o)$. Cases considered include: (i)

First-order upwinding; (ii) Lax-Wendroff (Leith); (iii) Second-order upwinding; (iv) Fromm; (v) Quasi-third-order upwinding; (vi) Fully third-order upwinding (QUICKEST). Diffusion terms are included in a consistent manner.

DR. CHIN-SHUN LIN, Iowa State University: "Similar Solutions for Viscous Hypersonic Flow over a Slender 3/4 Power Body of Revolution"

For hypersonic flow with a shock wave, there is a similar solution consistent throughout the viscous and inviscid layers along a very slender 3/4-power body of revolution if the perfect-gas assumption is made. The strong pressure interaction problem can then be treated by the method of similarity. In the present study, numerical calculations are performed in the viscous region with the edge pressure distribution known from the inviscid similar solutions. The compressible laminar boundary-layer equations are transformed into a system of ordinary differential equations. The resulting two-point boundary value problem is then solved by the Runge-Kutta scheme with modified Newton's method for the corresponding boundary conditions. The effects of wall temperature mass bleeding, and body transverse curvature are investigated. The induced pressure, displacement thickness, skin friction, and heat transfer due to above effects are analyzed and estimated.

DR. CHARLES L. MERKLE, Pennsylvania State University: "Computation of Low Mach Number Flows"

The convergence of typical time dependent algorithms slows down or ceases entirely at low Mach numbers. Stability theory is used to demonstrate that for implicit schemes this is the result of approximate factorization errors, while for explicit schemes it is because of time step limitations. Two methods for circumventing this slowdown are demonstrated. Matrix preconditioning is effective to Mach numbers of 10^{-3} to 10^{-5}, below which it fails because of machine roundoff. A perturbation approach is effective for even lower speeds. Both of these techniques achieve Mach number independent convergence by controlling the eigenvalues, but it is also shown that direct implicit methods do not require eigenvalue control. Extension to low Reynolds number viscous flows is seen to introduce additional requirements on the low Mach number schemes. The techniques are demonstrated by application to low speed flows with strong heat addition. Emphasis is placed on understanding the physics and mathematics that cause these difficulties and their resolution.

DR. ASHER A. RUBINSTEIN, Tulane University: "Crack Path Trajectory Analysis"

The aim of this project is to evaluate dominating parameters controlling a crack path trajectory formation. The experimental trajectory was obtained by positioning a hole in the vicinity of a crack path. In order to perform the corresponding analysis, a numerical approach for representing a curvilinear crack has been developed and applied to the experimentally obtained crack trajectories. Specific attention was given to variation in major fracture mechanics parameters like stress intensity factor and energy release rate, due to crack extension along the given path and possible crack path variations.

DR. JAMES N. SCOTT, University of Dayton: "Numerical Simulation of Unsteady Flow in Axisymmetric Jet Shear Layers"

Unsteady flow in axisymmetric jet shear layers is computed by solving the time-dependent compressible Navier-Stokes equations. The computed results are used to examine the influence of jet Mach number and temperature on the production and interaction of vortex structures in the shear layer. The computations are carried out for jet Mach numbers of 0.3 and 0.8 with jet total temperatures up to 800 K. The influence of acoustic pressure excitation and temperature excitation on the vortex behavior are also investigated. For certain excitation amplitudes and frequencies regular periodic pairing of vortices is observed. The computed results generally agree with experimental observations. In addition the effect of grid refinement on the unsteady flow features is also studied.

DR. LEON van DOMMELEN, Florida State University: "Lagrangian Procedures for Unsteady Separation"

Random walk vortex simulations of the incompressible two-dimensional Navier-Stokes equations are presented. The followed procedure allows inclusion of considerably more vortices than previously possible. The results demonstrate some of the advantages and weaknesses of the random walk method. Possible implications for unsteady separation, such as the need for less emphasis on interactive methods and more on stability are discussed.

DR. BRAM van LEER, University of Michigan: "Recent Developments in Computing Euler and Navier-Stokes Solutions with Finite-Volume Codes"

Among finite-volume codes for the Euler equations the ones most amenable to analysis are based on the "projection-evolution" philosophy. In the projection phase the discrete solution, consisting of cell-averaged data, is interpolated in order to obtain cell-interface data. The interpolation is monotonicity-preserving, a necessary requirement for avoiding numerical oscillations. The interpolants in adjacent cells generally do not match at the interface. At the start of the evolution phase the two interface state vectors are blended into one vector of normal fluxes by the numerical flux formula. A time-stepping or relaxation scheme then advances the solution in time or toward a steady state. Navier-Stokes codes can easily be derived from such Euler codes by including central-difference approximations to the viscous and conductive transport terms. The three cornerstones of such codes, i,e., interpolation routine, inviscid flux formula, and marching algorithm, are discussed and compared to related components of other flow codes. Results are shown for two and three-dimensional flows, steady and unsteady, inviscid and viscous.

DR. J. D. A. WALKER, Lehigh University: "Effects of a Hairpin Vortex Convected in a Shear Flow"

The hairpin vortex is often suggested to be a basic building block in the structure of the time-dependent turbulent flows near walls. Experiments suggest that such vortices are able to reproduce themselves in a variety of ways. In particular, moving hairpin vortices are observed to actuate the creation of second hairpin vortices through a complex viscous-inviscid interaction with the flow near solid walls. This process appears to be the fundamental regenerative method of production wherein new turbulence is continually introduced into the outer region of the boundary layer. The present computational study is an investigation of this phenomenon. The objective is to understand the features of the motion of hairpin vortices in a shear flow, the nature of the viscous flow induced near the wall by the moving hairpin and how hairpin vortices interact with one another.

DR. YAU SHU WONG, University of Alberta: "Approximate Polynomial Preconditioning Techniques of Vector Supercomputers"

The conjugate gradient algorithm in conjunction with polynomial preconditioning has been proved to be very efficient for solving large systems of linear equations on vector supercomputers. Unlike the standard polynomial preconditioning which is based on a truncated polynomial expansion using the original matrix operator, we introduce an approximate polynomial preconditioning strategy. This technique used the fact that a simple but effective polynomial preconditioning may be constructed via another coefficient matrix and, a particularly attractive feature of this approach is that the computational cost for the preconditioning step can be substantially reduced. Numerical solutions for solving three-dimensional second-order elliptic partial differential equations and two-dimensional fourth-order equations are given to demonstrate the effectiveness of this method.

OTHER ACTIVITIES

A workshop was instituted to familiarize visiting researchers with the Lewis computing facilities. The workshop convened every morning during the summer months to solve problems and answer questions. Also to acquaint the ICOMP staff with pertinent Lewis research programs and goals, overviews were given of the programs in the Internal Fluid Mechanics and Structures Divisions and the Office of the Chief Scientist.

Another important activity was the organization of an Advisory Committee to review the ICOMP program and make recommendations for future directions for ICOMP. The committee members are:

Dr. Ted Belytschko, Northwestern University

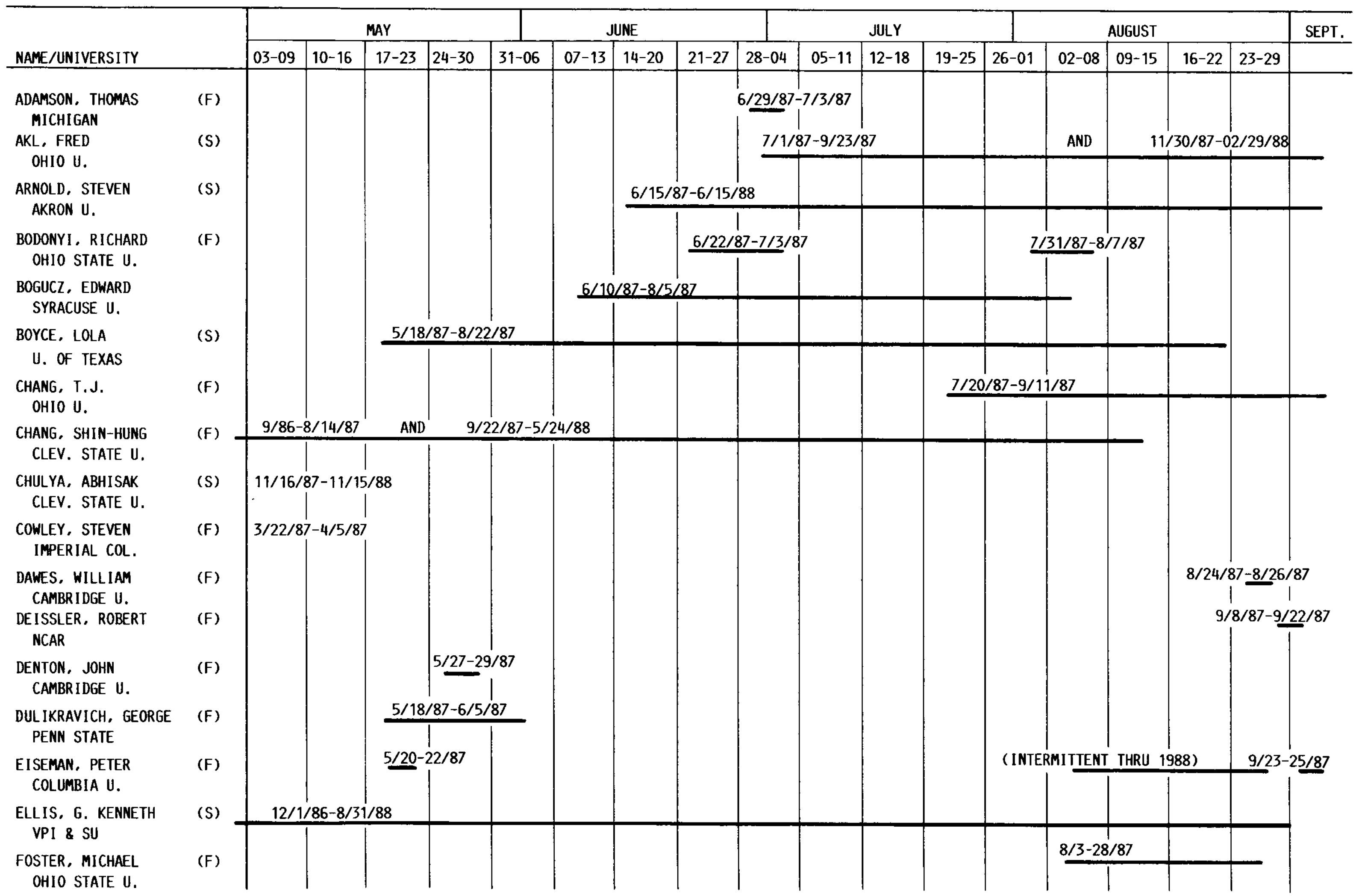
Dr. Earll Murman, M.I.T.

Dr. J. Tinsley Oden, University of Texas

Dr. Steven Orszag, Princeton University

Dr. Stanley Rubin, University of Cincinnati

This committee met in November, 1987 at Lewis. Following a review of the ICOMP program, committee members interviewed ICOMP and Lewis personnel and then formulated recommendations. Among them was the recommendation that ICOMP select one or two areas each year to be given special emphasis by organizing appropriate seminars and colloquia and by selecting visiting researchers well-known in the area. ICOMP expects to make a concerted effort to accomplish this in 1988. Other recommendations are under consideration and, where feasible, will be incorporated into the future efforts of ICOMP.



(S) STRUCTURES (F) FLUID MECHANICS

FIGURE1. - ICOMP VISITING RESEARCHERS, 1987.

NAME/UNIVERSITY		DATES
GUPTA, MURLI GEOR. WASH. U.	(F)	8/3/87-8/31/87
HAGSTROM, T. SUNY, STONY BR.	(F)	6/8/87-7/31/87
HARIHARAN, S.I. UNIV. OF AKRON	(F)	10/8/87-5/24/88
JIANG, BO-NAN U. OF TEXAS	(F)	10/19/87-10/18/88
KORCZAK, KAROL CWRU	(F)	5/26/87-8/24/87
LEE, SHIH-TUEN NAT. TAIWAN	(F)	7/1/86-8/31/87
LEONARD, B.P. AKRON U.	(F)	5/18/87-7/25/87 AND 9/15/87-5/24/88
LIN, CHIN-SHUN IOWA STATE	(F)	8/1/87-7/31/87
LIU, JONG-SHANG CWRU	(F)	6/1/87-5/31/88
MEISTER, JEFFREY AKRON U.	(S)	10/1/87-12/31/87
MERKLE, CHARLES PENN STATE U.	(F)	7/13/87-7/24/87
MESSITER, ARTHUR MICHIGAN	(F)	6/29/87-7/3/87
PERIDIER, VALLORIE LEHIGH U.	(F)	6/29/87-7/17/87
RASHIDI, MAJID CWRU	(S)	4/1/87-3/31/88
RUBINSTEIN, A TULANE U.	(S)	6/1/87-8/3/87
SARIGUL, NESRIN OHIO U.	(S)	6/19/87-7/4/87 8/31/87-9/4/87
SCHAFFER, STEVEN BEREA COLLEGE	(F)	5/26/87-7/24/87

FIGURE 1. - CONTINUED.

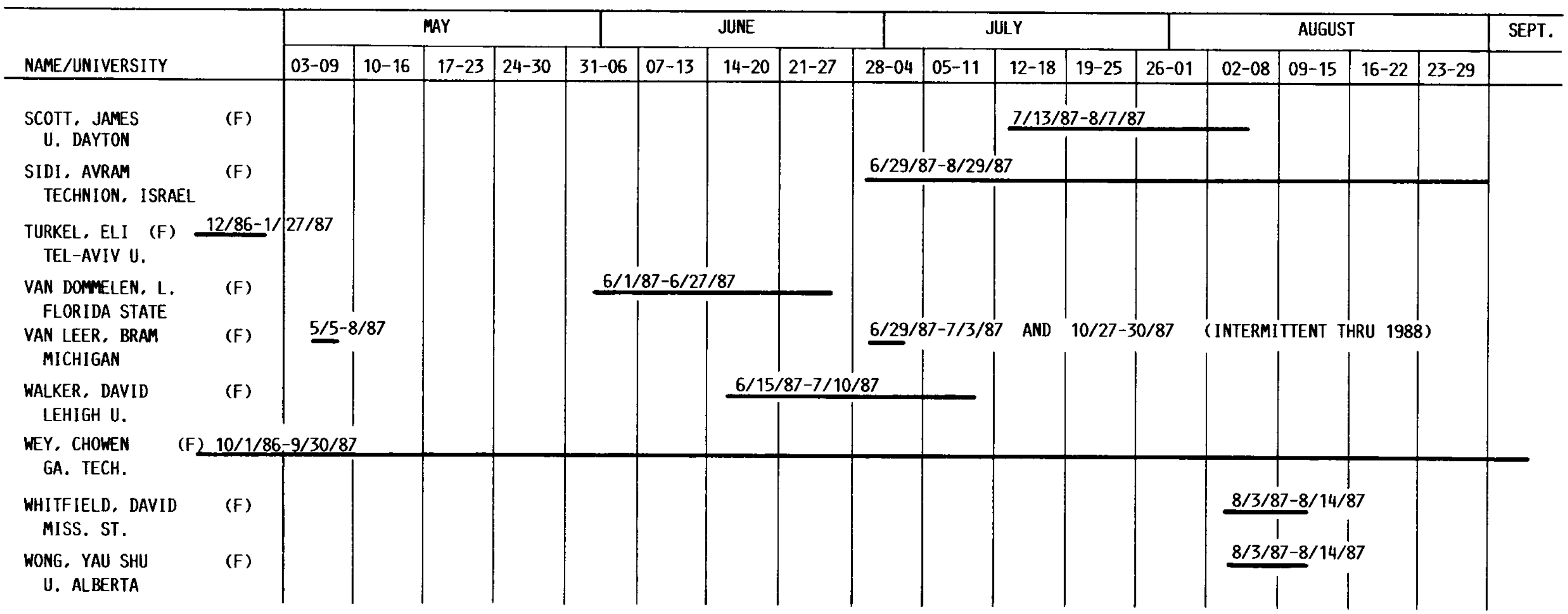


FIGURE 1. - CONCLUDED.

ORIGINAL PAGE IS
OF POOR QUALITY

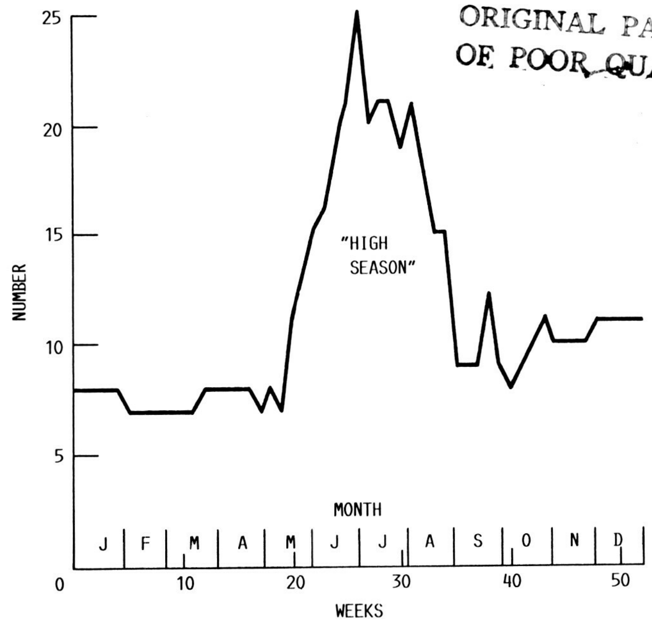


FIGURE 2. - 1987 ICOMP WEEKLY RESIDENT STAFF.

FIGURE 3. - ICOMP STEERING COMMITTEE AND VISITING RESEARCHERS IN JUNE, 1987. 1ST ROW SEATED (LEFT TO RIGHT): CHOWEN WEY, NESRIN SARIGUL, LOLA BOYCE, DARLEEN MIDKIFF, FRANCES PIPAK, SHEILA NUSSBAUM, ELI RESHOTKO. 2ND ROW: JONG-SHANG LIU, LOU POVINELLI, JEFFREY MEISTER, ISAAC GREBER, RICHARD BODONYI, LEON VAN DOMMELEN, MAJID RASHIDI, GRAHAM ELLIS, B.P. LEONARD, FRED AKL, JOHN KLINEBERG, CHARLES FEILER. 3RD ROW: BRENT MILLER, LESTER NICHOLS, DAVID WALKER, THOMAS HAGSTROM, ASHER RUBINSTEIN, EDWARD BOGUCZ, STEVEN ARNOLD, STEVEN SCHAFFER, CHIN-SHUN LIN, FRED CHANG, KENNETH PEDERSEN.

UNIVERSITY OR INSTITUTION	NUMBER
1. ALBERTA-EDMONTON	1
2. AKRON	4
3. CAMBRIDGE	2
4. CASE WESTERN RESERVE	3
5. CLEVELAND STATE	2
6. COLUMBIA	1
7. DAYTON	1
8. FLORIDA STATE	1
9. GEORGE WASHINGTON	1
10. GEORGIA TECH	1
11. IMPERIAL COLLEGE	1
12. IOWA STATE	1
13. LEHIGH	2
14. MICHIGAN	3
15. MISSISSIPPI STATE	1
16. NATIONAL TAIWAN	1
17. NEW MEXICO INSTITUTE OF TECH	1
18. OHIO	2
19. OHIO STATE	3
20. PENN STATE	2
21. SUNY-STONY BROOK	1
22. SYRACUSE	1
23. TECHNION-ISRAEL INSTITUTE OF TECH.	1
24. TEL-AVIV	1
25. TEXAS	2
26. TULANE	1
27. VPI & SU	1
28. NCAR	1
	43

FIGURE 4. - COMPOSITION OF 1987 STAFF - INSTITUTIONS REPRESENTED.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA TM-100790 ICOMP-88-1	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Institute for Computational Mechanics in Propulsion (ICOMP) Second Annual Report - 1987		5. Report Date March 1988
		6. Performing Organization Code
7. Author(s)		8. Performing Organization Report No. E-3964
		10. Work Unit No. 505-62-21
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001		14. Sponsoring Agency Code

15. Supplementary Notes

Report produced by the Steering Committee, Institute for Computational Mechanics in Propulsion, NASA Lewis Research Center (work funded under Space Act Agreement C99066G).

16. Abstract

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1987.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Numerical analysis Computer science Mathematics	Unclassified - Unlimited Subject Category 64

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No of pages	22. Price*
Unclassified	Unclassified	46	A03

NASA FORM 1626 OCT 86

*For sale by the National Technical Information Service, Springfield, Virginia 22161